GSK980TD Turning CNC

PLC User Manual

广州数控设备有限公司
GSK CNC EQUIPMENT CO., LTD.

Book 1 PROGRAMMING

PLC specification, PLC address, ladder

Book 1 Programming

Book 2 FUNCTION

PLC function control logic and relative signals

Book 2 Function

Book 3 CNC CONFIGRUTION SOFTWARE

GSKCC software and usage

Book 3 CNC Configuration software

APPENDIX

Appendix

BOOK 1

PROGRAMMING

Contents

Chapter 1　SEQUENTIAL PROGRAM

1.1 PLC SPECIFICATION

For different CNC PLC, there are different program capacity, processing speed, function instructions and nonvolatile memory addresses. Specifications of GSK980TD PLC are as follows:

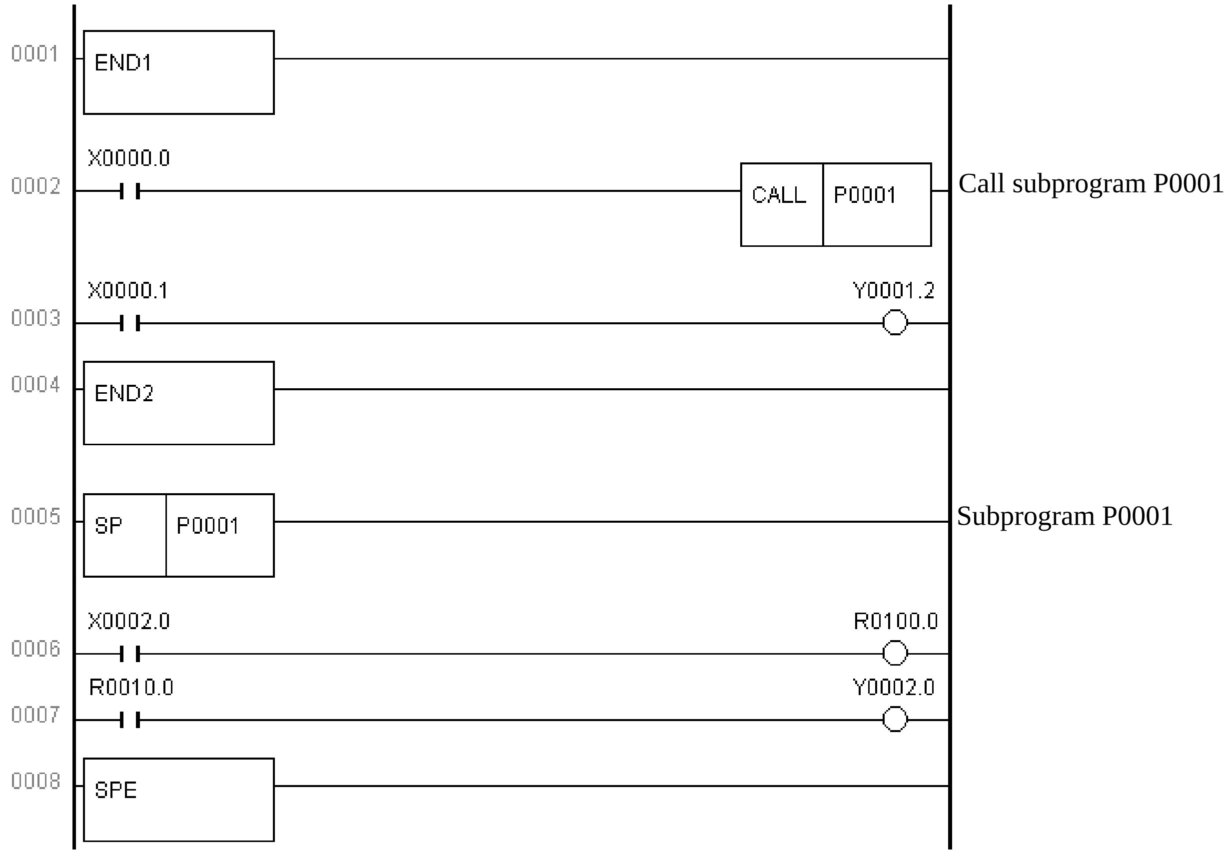
Specification		980TD-PLC
Programming language		Chinese ladder
Programming software		GSKCC.exe
Programming grades		2
Executive cycle of grade one program		8ms
Average processing time of basic instruction		<2μs
Max. step of program		5000 steps
Programming instruction		Basic instruction +function instruction
Programming address	**Internal relay address（R）**	R0000～R0999
	Information displaying request address（A）	A0000～A0024
	Timer address （T）	T0000～T0099
	Counter address（C）	C0000～C0099
	Data tablet address（D）	D0000～D0999
	Hold relay address（K）	K0000～K0039
	Counter preset value address（DC）	DC0000～DC0099
	Timer preset value address（DT）	DT0000～DT0099
	Subprogram address（P）	P0000～P9999
	Label address （L）	L0000～L9999
	Machine→PLC address（X）	X0000～X0029
	PLC→machine address（Y）	Y0000～Y0019
	CNC→PLC address （F）	F0000～F0255
	PLC→CNC address（G）	G0000～G0255

1.2 STRUCTURE of SEQUENTIAL PROGRAM

Sequential program is defined to logically control the machine and relative devices according to sequence of ladder. The sequence of ladder is compiled in a traditional PLC but GSK980TD CNC PLC is integrated by traditional one and advanced structured programming, employed with subprogram, embedded subprogram and conditional branch，and with apparent advantages compared to the traditional.

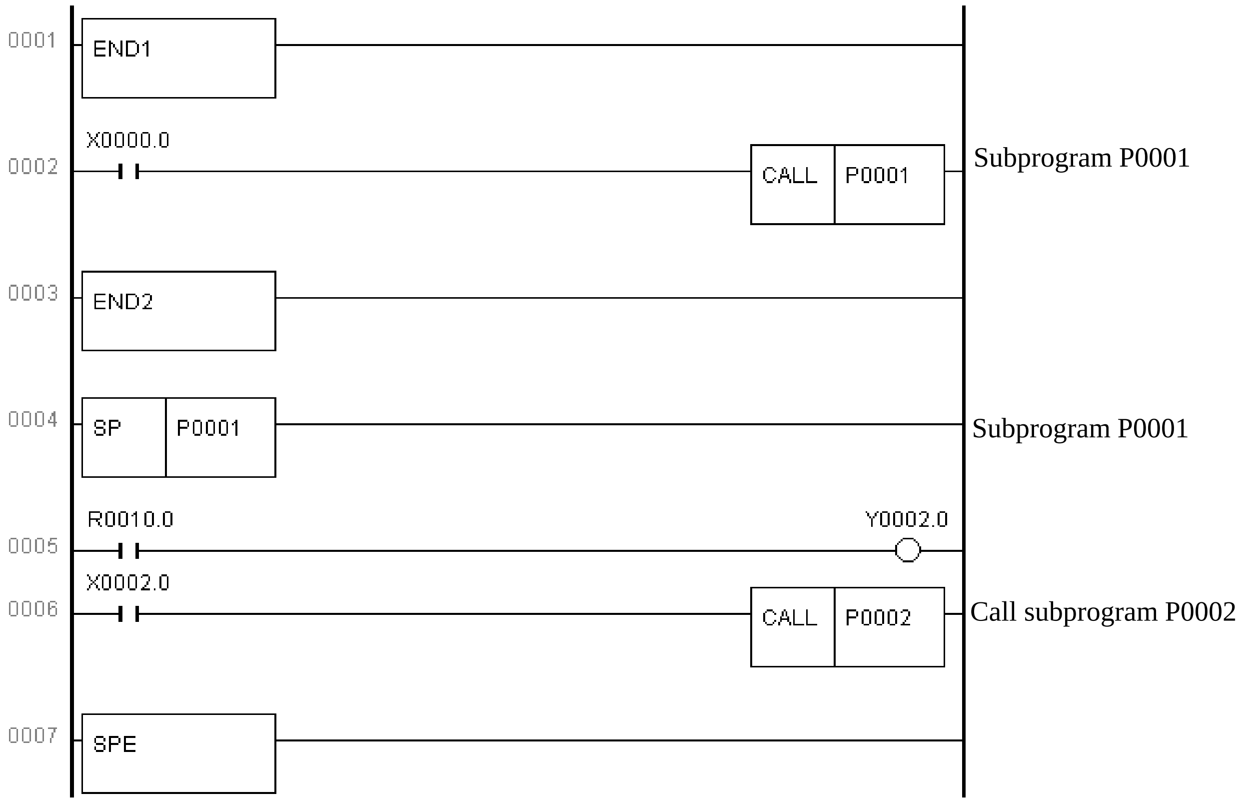
1.2.1 Subprogram

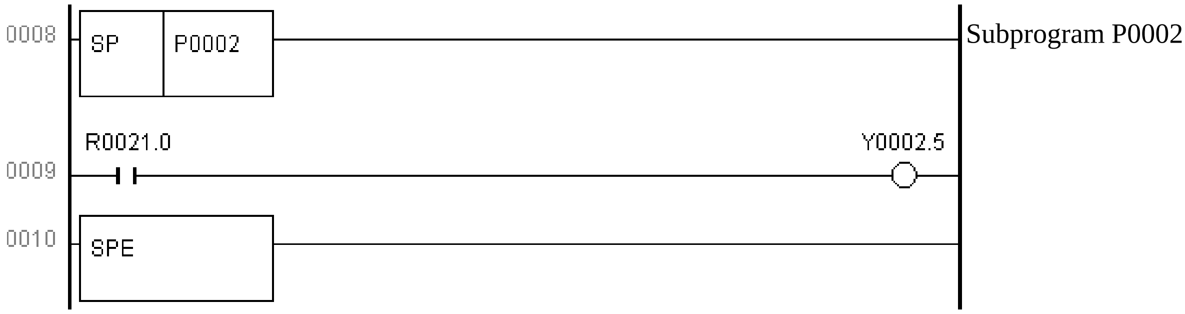
In GSK980TD CNC PLC program, a special subprogram can be called according to requirements of programming. For example, subprogram P0001 can be called when contactor X0000.0 is closed as follows:



1.2.2 Subprogram Embedding

GSK980TD CNC PLC can realize 20 grades subprogram embedding as the following figure. Subprogram P0001 can be called when contractor X0000.0 is closed in main program; subprogram P0002 can be called when contractor X0002.0 is closed in the subprogram P0001.





1.2.3 Conditional Branch

The system can judge whether the setting conditions to execute a corresponding subprogram are satisfactory when a main program is executed circularly, otherwise the system executes subprograms in order as the above-mentioned figure.

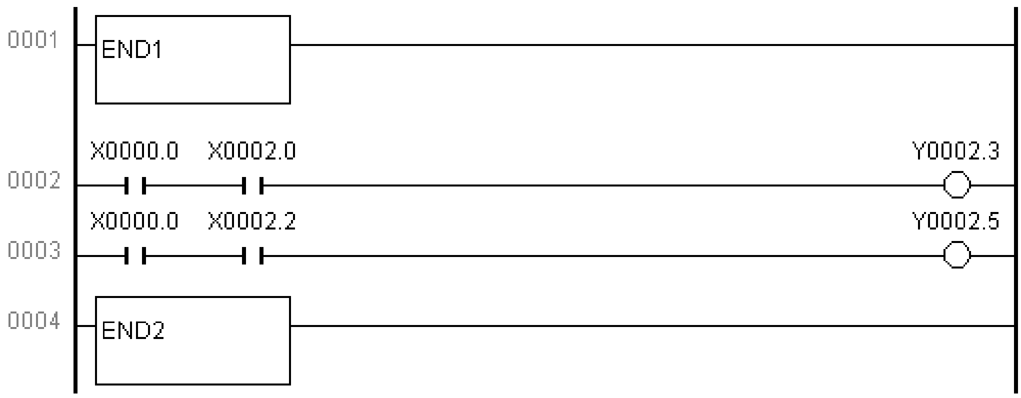
1.3 EXECUTING SEQUENTIAL PROGRAM

The edited sequential programs(ladder program)are downloaded to 980TD by serial, CNC reads the ladder and converts into some format identified by it after it is switched on again, and then CPU decodes and operation processes them to store into RAM, and last reads every instruction in memory to execute it by arithmetical operation.

1.3.1 Execution Process of Sequential Program

PLC sequence control is executed by software and there is different from general relay circuit, and so its method is understandingly considered in editing PLC sequential programs.

Every relay can output simultaneously for general relay control circuit as the following figure. Y0002.3 and Y0002.5 output simultaneously when contactor X0000.0, X0002.0 and X0002.2 are closed; in PLC sequence control, every relay outputs in order. For example, Y0002.3 outputs and then Y0002.5 does when X0000.0, X0002.0 and X0002.2 are closed, namely, outputs are executed in order as ladder.

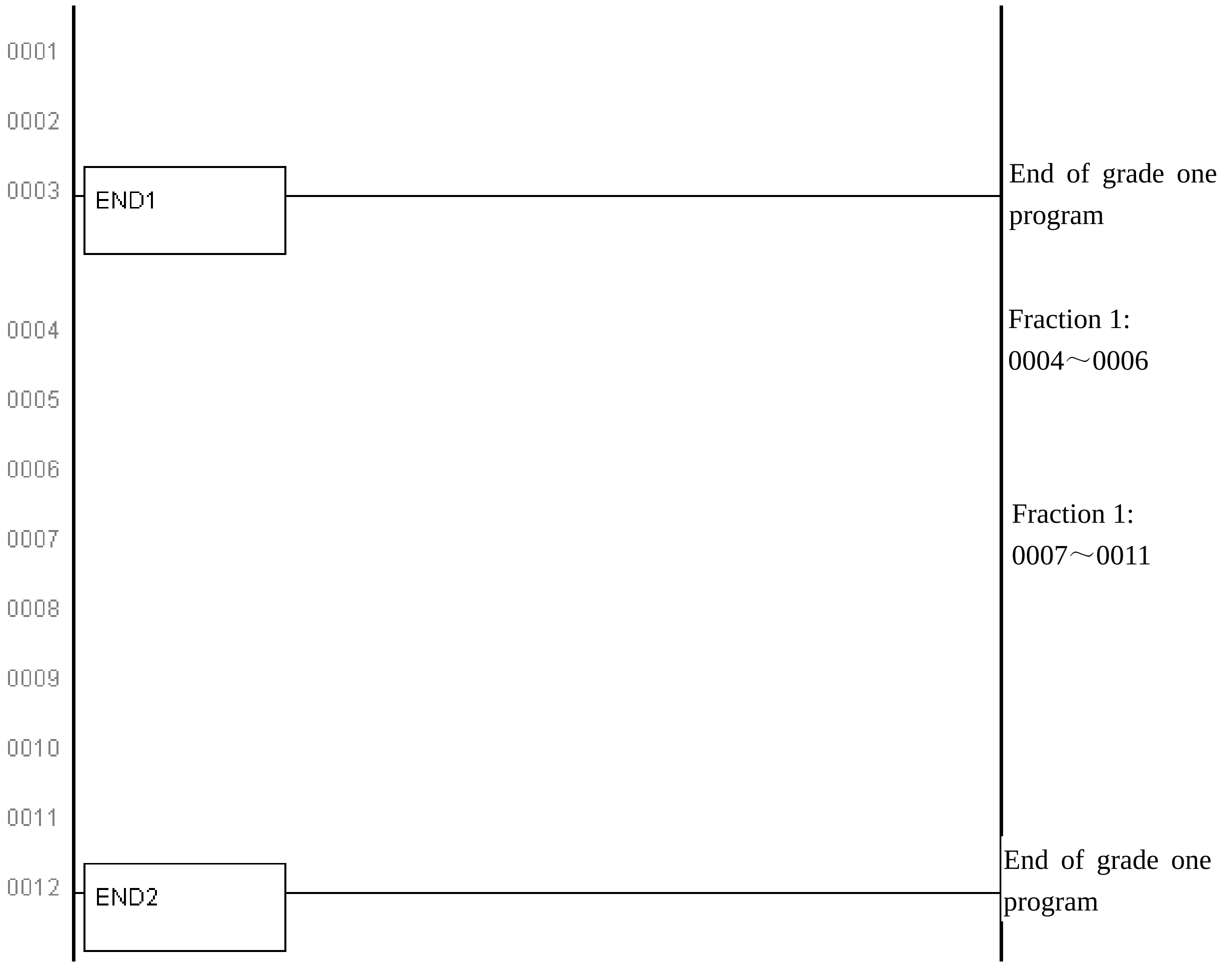


1.3.2 Cycle Execution of Sequential Program

Cycle execution of sequential program is defined that PLC executes ladder from its home to end, and again from its home to end after the run is completed.
Processing cycle is defined to runtime of ladder from home to end. The shorter the processing cycle is, the stronger the response of signal is.

1.3.3 Priority Order of Executing Sequential Program

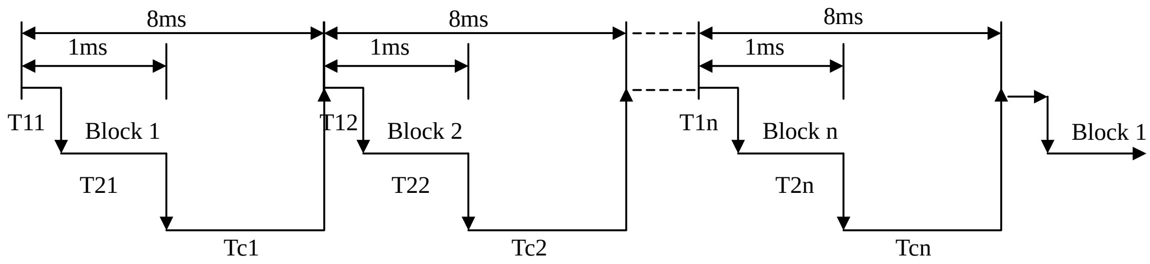
GSK980TD PLC programs are divided into grade one and grade two programs which processing cycles are different. The first one is executed per 8ms to do with short pulse signal with quick reaction, and the second one is executed per 8n(ms),in which n is fraction times. PLC divides the grade two program into n blocks according to runtime, and executes one block per each 8(ms).



Fraction of the grade two program is to execute the first and execution process is as the following figure when the fraction number is n; T11, T12, T1n are runtime to execute the first. T21, T22, T2n separately corresponds to runtime of No. n block in executing the second when No. n cycle is execute. Tc1, Tc2, Tcn separately corresponds to hold time by CNC when No. n cycle is executed.

The program returns to its beginning to continue execution when the last fraction block of the second is executed completely. The first is executed every 8ms, No. n block of the second is done every 8n, and runtime

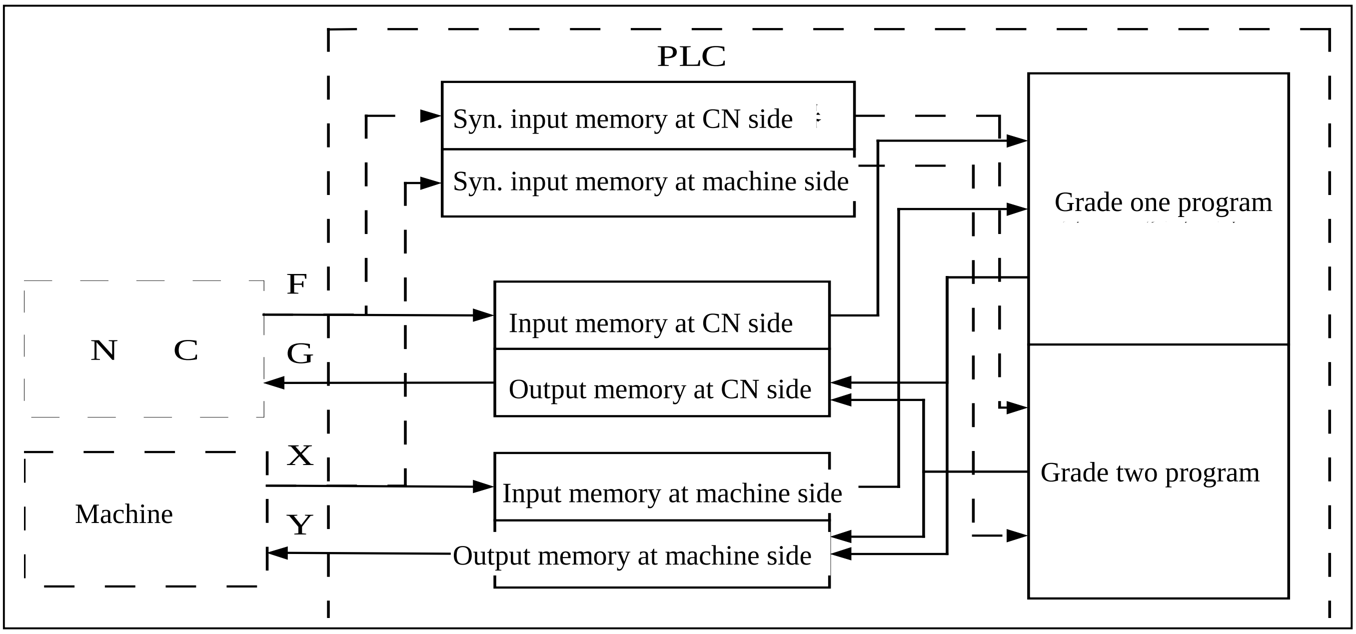
of one cycle is 8n(ms). The more the fraction number of the second is, the longer the runtime of one cycle is.



1.4 PROCESSING INPUT/OUTPUT SIGNAL

Processing of input/output signal is as the following figure. X signal of machine I/O interface and F signal of NC are separately input to input memories at machine side and NC side, and directly used by the first grade program; they separately input to synchronous input memories are used by the second. Output signals of the first and the second are separately output to output memories at NC side and machine side, and then separately output to NC and I/O interface of machine

Signal states of the above-mentioned memories are displayed by diagnostic interface, and the diagnostic number corresponds to address number of program.



1.4.1 Processing Input Signal

A: Input signal of grade one program:

Input memory at NC side is scanned every 8ms and stores F signal from NC, and the system directly use its state when the first is executed.

B: Input signal of grade two program:

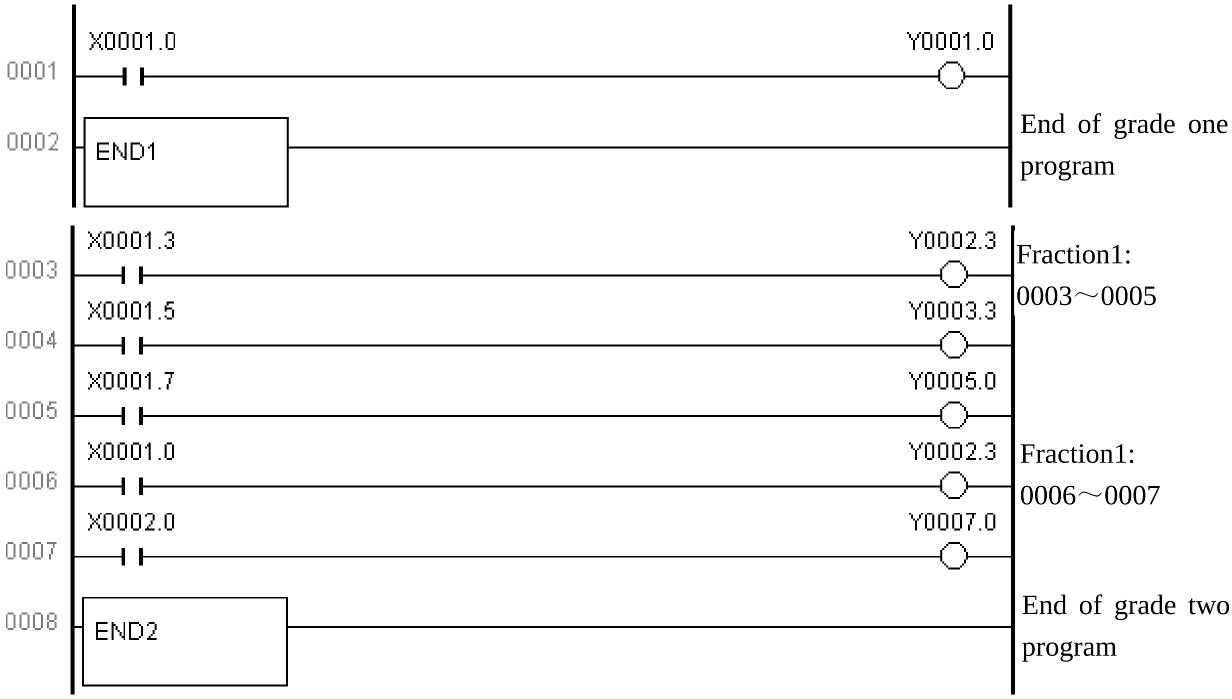
Input signal of the second is the one stored by the first. The first directly use F and X signal and so the input

signal of the second lags the first one and its max. lag time is runtime of one grade two program.

C：Difference of input signal states of the first and the second:

For the same input signal, its states are different in the first and the second when PLC reads input signals, because the first reads input memories at NC side and machine side but the second reads the synchronous input memory at NC side and ones at machine side. The input signal of the second lags behind that of the first and its lag time is 8nms which should be noted.

Example：



When the lag time is the first 8ms, X0001.0=1 and the system executes the first Y0001.0=1. When the system starts to execute the second, X0001.0＝1 inputs to the synchronous input memory and starts to execute the first block of the fractional second.

When the lag time is the second 8ms, X0001.0=0 and the system executes the first one Y0001.0=1. And then the system executes the second block of the fractional second but X0001.0 =1 which state is still the previous synchronous input memory and Y0002.3=1 after execution.

1.4.2 Processing Input Signal

A：Signal to NC

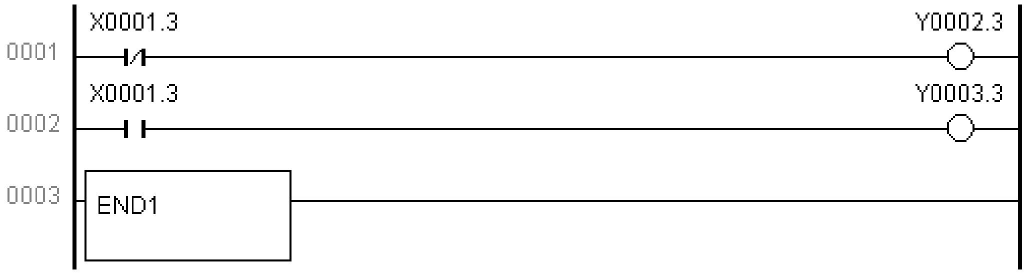
PLC transmits output signal to the output memory at NC side every 8ms, and then directly outputs to NC.

B：Signal to machine

PLC transmits output signal to the output memory at machine side, and then directly outputs to memory every 2ms.

1.4.3 Synchronous Procession of Short Pulse Signal

The first is used for processing the short pulse signal. When it is less than 8ms, namely when the system executes the first, the input signal state may be changed, which may execute programs by mistake.



End of grade one program

As above, X0001.3=0 is changed to X0001.3=1 after Y0002.3=1 is executed, and if the system executes the next line of ladder and Y0003.3=1, at the moment Y0002.3=1 and Y0003.3=1. To avoid the above, process synchronously the short pulse signal as follows:



End of the first grade program

After the program is executed synchronously and when X0001.3=1，Y0003.3=1，Y0002.3=0， X0001.3=0，Y0002.3=1，and Y0003.3=0，but Y0003.3=1 or Y0002.3=1.

1.4.4 Interlock Signal

For safety, the signals must be employed with soft interlock in sequence control, and with hard interlock to relay control circuit of power electric box at machine side at the same time. Because the hardware is failure, the interlock is invalid in executing sequential program even if it is employed logically with soft interlock, which can ensure the operator is not injured and the machine is prevented from damage.

1.5 EDITING SEQUENTIAL PROGRAM

Edit the sequential program from ladder which is composed of relay contacts, symbols and function instructions. Logic relationship in ladder consists of sequential program which is edited by two methods: one is employed with input to use program instructions and the other is with relay symbols by their corresponding contactor, symbol and function instruction. Edit the sequential program employed with the ladder format instead of mnemonic code language when the system is employed with the relay symbol.

In actual editing sequential program, use programming instruction or ladder to edit it according to PLC. In the User Manual, the system is employed with ladder as follows:

1.5.1 Distributing Interface (Step 1)

The interfaces can be distributed after control target is defined and the corresponding input/output signal points are counted. Refer to input/output interface signal list

1.5.2 Editing Ladder (Step 2)

Edit the software GSKCC.EXE by 980TD ladder to represent the required control operations of machine. Counter, timer which are not done by relay symbol is represented by the specified function instructions. The edited ladder is downloaded to CNC by serial.

1.5.3 Debugging Ladder (Step 3)

After the ladder is downloaded to 980TD, the ladder is debugged as follows:

A： emulator

Use one emulator instead of machine to debug it. Machine signal state is represented with switch ON/OFF, and output signal state is done with indicator ON/OFF. Observe if every indicator on the emulator is correct when executing CNC.

B： CNC diagnosis

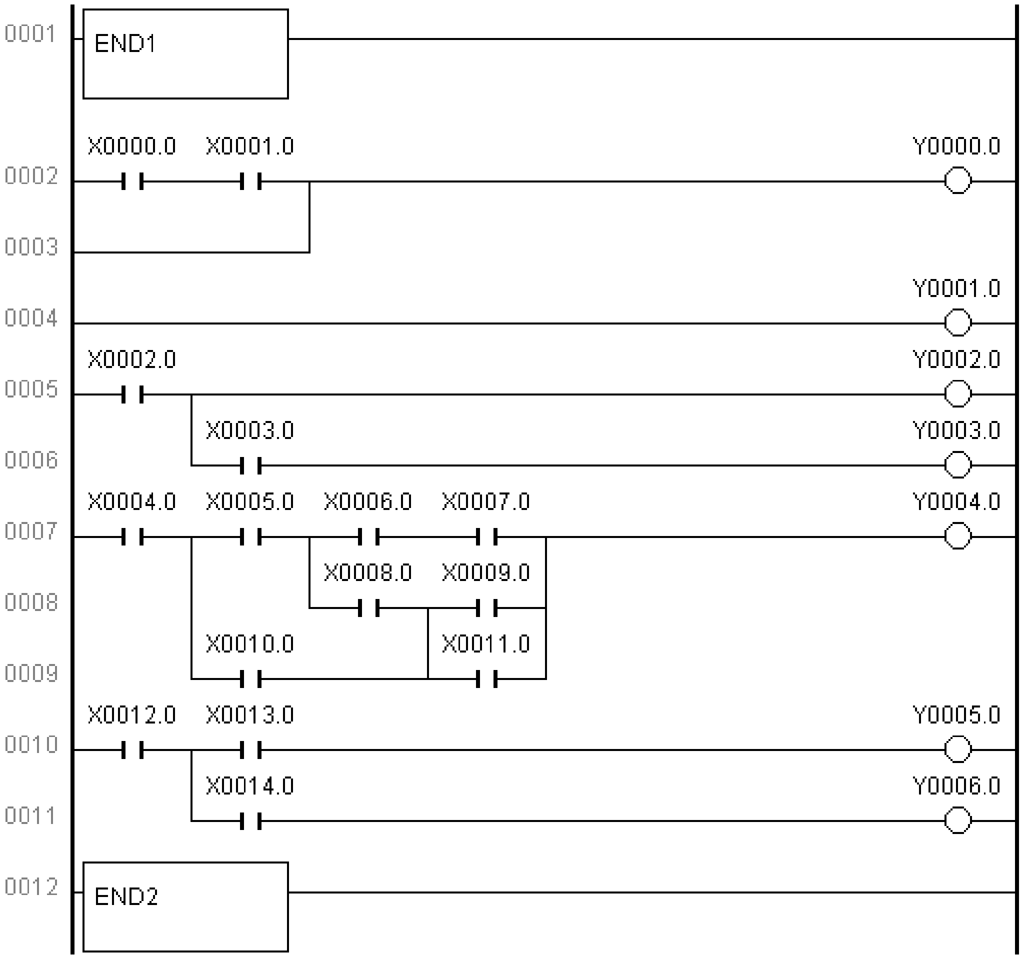
Observe if the diagnostic state of every signal is consistent with the function requirement when executing CNC. Check the ladder by checking each function in order.

C： actual run

There may be an unexpected result in the actual debugging machine and so do preventive measures before debugging.

1.5.4 Program Editing Limit

In program, END1 and END2 are needed, which are taken separately as ending character of the first and the second, and END1 must be before END2. The system only supports parallel output instead of multi grades output and there are syntactic errors as follows:



End of grade one program

Syntactic error: 0002～0003

Syntactic error: 0004

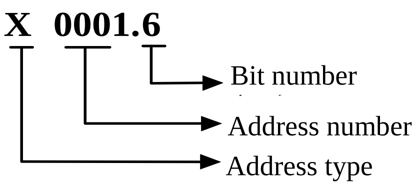
Syntactic error: 0005～0006

Syntactic error: 0007～0009

Syntactic error: 0010～0011

Chapter 2 ADDRESS

Addresses are used for distinguishing signals. Different addresses separately correspond to input/output signal at machine side and CNC side, internal relay, counter, timer, holding relay and data list. An address number is consisted of address type, address number and bit number as follows:



Address type: X, Y, R, F, G, K, A, T, DT, DC, C, D, L, P
Address number: decimal number to express one byte
Bit number: octal number, 0～7 separately expressing byte 0～7 bit of front address number

980TD PLC addresses are divided into fixed addresses and definable addresses. Signal definitions of the fixed addresses cannot be changed and are defined by CNC; the definable addresses can be defined again by user according to the actual requirements. Address types are as follows:

Address	Explanation	Range
X	Machine→PLC	X0000～X0029
Y	PLC→machine	Y0000～Y0019
F	NC→PLC	F0000～F0255
G	PLC→NC	G0000～G0255
R	Intermediate relay	R0000～R0999
D	Data register	D0000～D0999
C	Counter	C0000～C0099
T	Timer	T0000～T0099
DC	Counter preset value register	DC0000～DC0099
DT	Timer preset value register	DT0000～DT0099
A	Information displaying request signal	A0000～A0024
K	Hold relay	K0000～K0039
L	Jump label	L0000～L9999
P	Subprogram label	P0000～P9999

Note: address R900～R999, K30～39 are used for reserved area of CNC program instead of output relay.

2.1 MACHINE→PLC ADDRESS（X）

980TD PLC X addresses are divided into two types,the first one is X0000.0～X0003.7,which are mainly distributed to CNC XS40 and XS41 I/O interfaces, including fixed addresses and definable addresses,and the second one is X0020.0～X0026.7,which are fixed addresses and mainly be distributed to input keys on operator panel. Other addresses are reserved ones. The value range is 0 or 1.

2.1.1 X Address in I/O Interface

- Address range: X0000.0～X0003.7 are separately distributed to CNC XS40 and XS41 I/O interface.
- Fixed address: X0000.3, X0000.5, X0001.3 separately corresponds to XDEC, ESP, ZDEC signal which can be directly distinguished by CNC in CNC run.

Example: ESP signal is can be connected to X0000.5, CNC directly distinguishes signals on it and judge if there is ESP signal; CNC alarms to emergently stop when G8.4 signal is valid by PLC control.

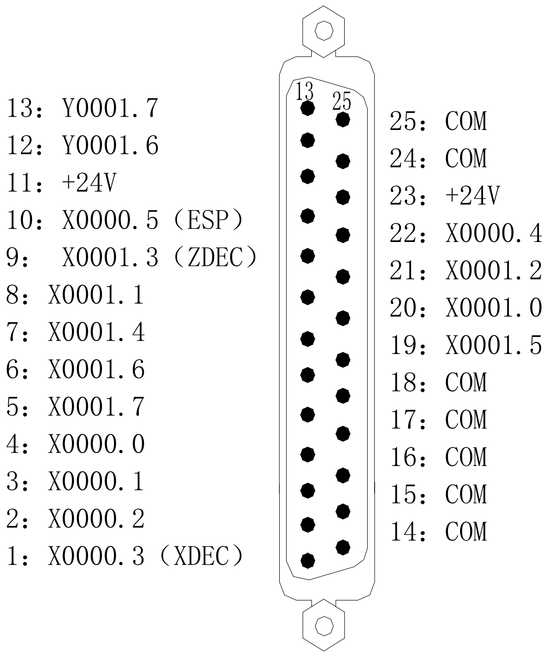
Namely: CNC alarms to emergently stop when X0000.5 is 0;

CNC alarms to emergently stop when G8.4 is 0 by PLC control.

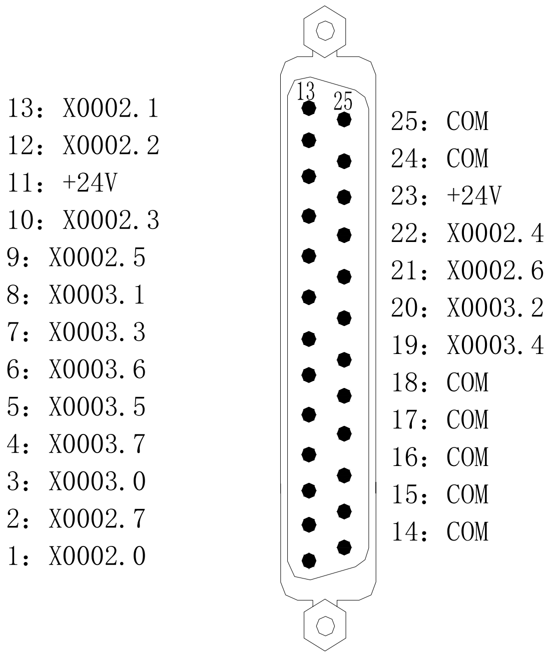
Input signal of fixed addresses

Signal	Symbol	Address
Emergent stop signal	ESP	X0000.5
Deceleration signal of machine reference point return in X direction	XDEC	X0000.3
Deceleration signal of machine reference point return in Z direction	ZDEC	X0001.3

- Definable address: their functions can be defined by user according to requirement and used for connecting with external electric circuit and ladder. Distribution graph of X address in I/O interface is as follows:



XS40（female）



XS41（female）

2.1.2 X Address on Operator Panel

Address range: X0020.0～X0026.0 are fixed addresses which correspond to press keys on operator panel and which signal definitions cannot be changed by user.

Relationship between addresses and press keys is as follows:

X0020	7	6	5	4	3	2	1	0
Key	SKIP	SINGLE	JOG	MPG	MACHINE ZERO	MDI	AUTO	EDIT

X0021	7	6	5	4	3	2	1	0
Key	X	0.1	0.01	0.001	PROGRAM ZERO	DRY	MST M. S. T. LOCK	MST M. S. T. LOCK

X0022	7	6	5	4	3	2	1	0
Key	Z		Z				Z	

X0023	7	6	5	4	3	2	1	0
Key	CCW	JOG LUR.	STOP	COOLANT	CW			

X0024	7	6	5	4	3	2	1	0
Key	PAUSE	%	%	%	%	%	%	TOOL

X0025	7	6	5	4	3	2	1	0
Key								RUN

X0026	7	6	5	4	3	2	1	0
Key								RESET

2.2 PLC→MACHINE ADDRESS（Y）

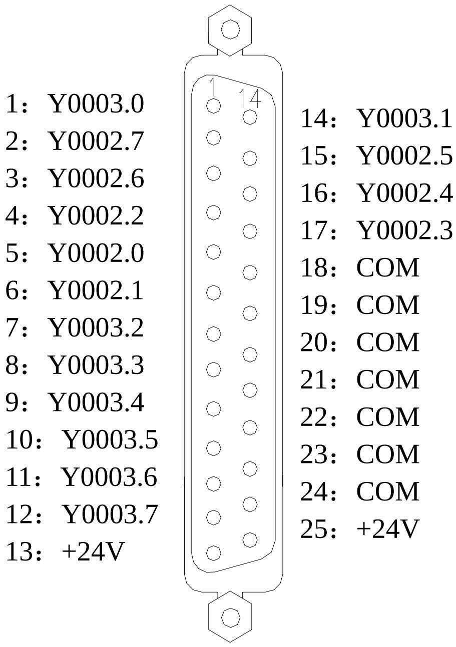
980TD-PLC Y address are divided into two: Y0000.0～Y0003.7 are mainly distributed to CNC XS42 and XS39 I/O interfaces, including fixed address and definable address; Y0004.0～X0009.7 which are mainly distributed to indicators on the operator panel. Other addresses are reserved ones. Their values are 0 or 1.

2.2.1 Y Address in I/O Interface

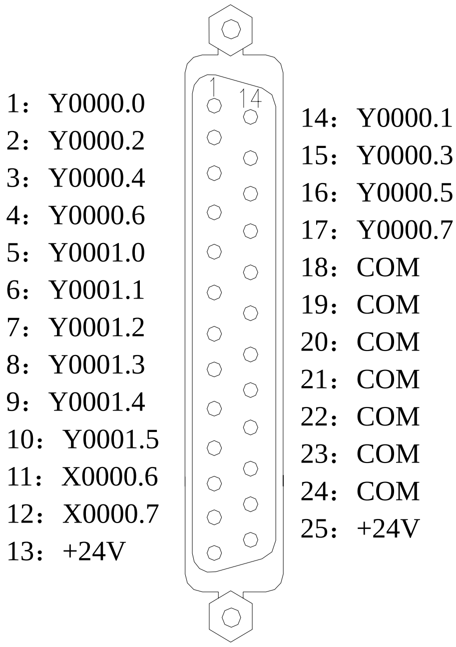
Address range: Y0000.0～Y0003.7 are separately distributed to CNC XS42 and XS39 I/O interfaces and their signal definitions can be defined by user according to requirements to connect to external

electric circuit and ladder.

Distribution graph of 980TD output interfaces address is as follows:



XS42（male）



XS39（male）

2.2.2 Y Addresses on Operator Panel

Address range: Y0004.0～Y0009.0 are fixed addresses which correspond to indicators on the operator panel, and which signal definitions cannot be changed by user. Relationship corresponding to each state indicator is referred to Appendix2: Output signal(Y).

2.3 PLC→NC ADDRESS（G）

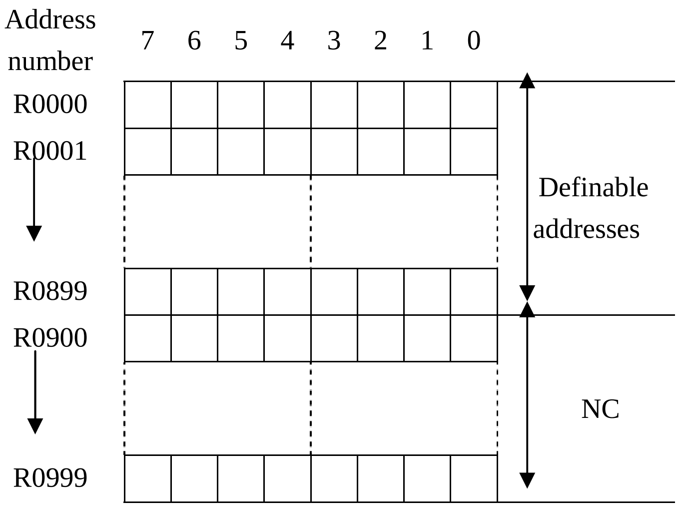
Address range: G0000.0～G0255.7, value range: 0 or 1. Refer to Appendix3: G, F signals about definitions of address signals.

2.4 NC→PLC ADDRESS（F）

Address range: F0000.0～F0255.7, value range: 0 or 1. Refer to Appendix3: G, F signal about definitions of address signals.

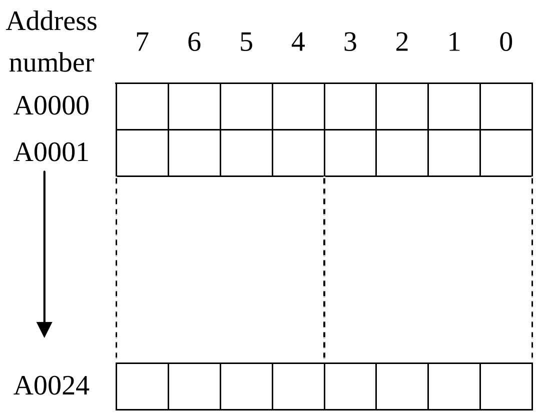
2.5 INTERNAL RELAY ADDRESS

Address range: R0000.0～R0999.7, value range: 0 or 1. They are zero after CNC is switched on.



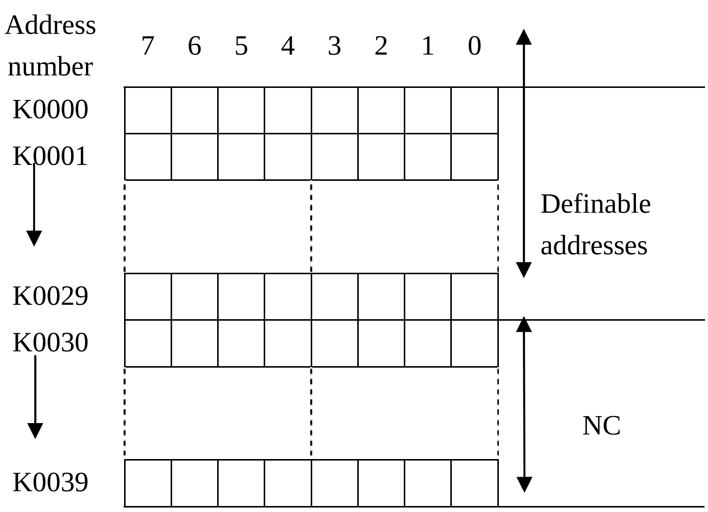
2.6 INFORMATION DISPLAYING REQUEST ADDRESS（A）

Address range: A0000.0～A00024.7 and they are zero after CNC is switched on.



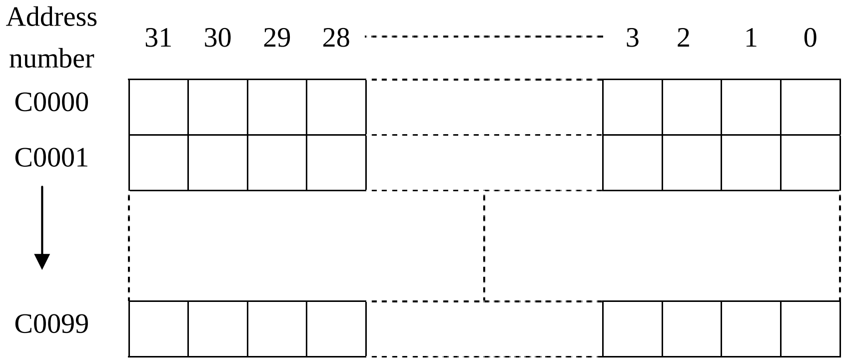
2.7 HOLD RELAY ADDRESS（K）

The address area is used for hold relay and setting PLC parameters and data are saved after the system is switched off. Address range: K0000.0～K0039.7, value range: 0 or 1.



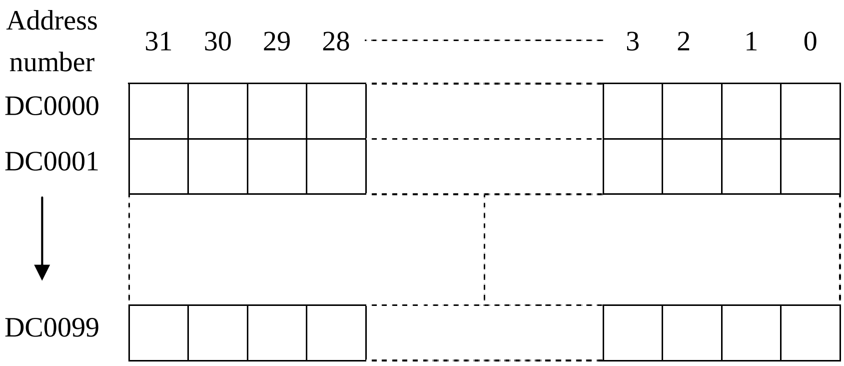
2.8 COUNTER ADDRESS（C）

The address area is used for storing current counting value of counter and data are saved after the system is switched off. Address range: C0000～C0099, value range: 0～21,4748,3647.



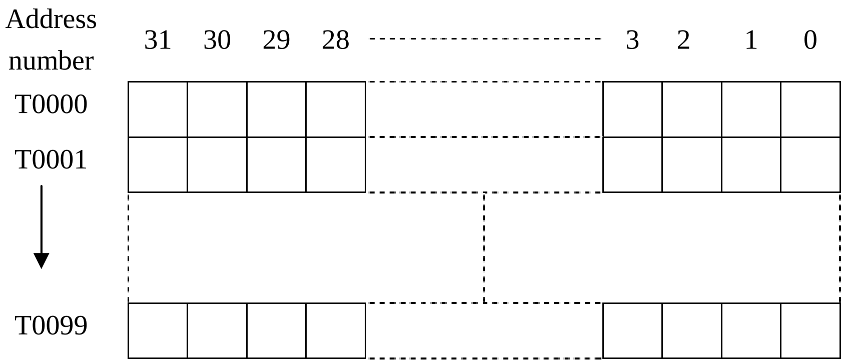
2.9 COUNTER PRESET VALUE ADDRESS（DC）

The address area is used for storing preset value of counter and data are saved after the system is switched off. Address range: DC0000～DC0099, value range: 0～21,4748,3647.



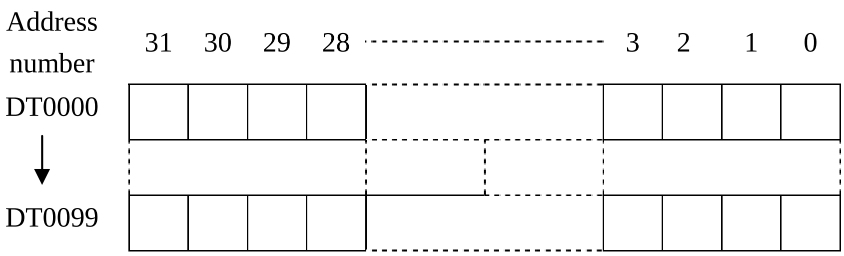
2.10 TIMER ADDRESS（T）

The address area is used for storing current value of timer and T0000～T0079 are zero after the system is switched on.T0080～T0099 are saved after it is switched off. Value range: 0～21,4748,3647.



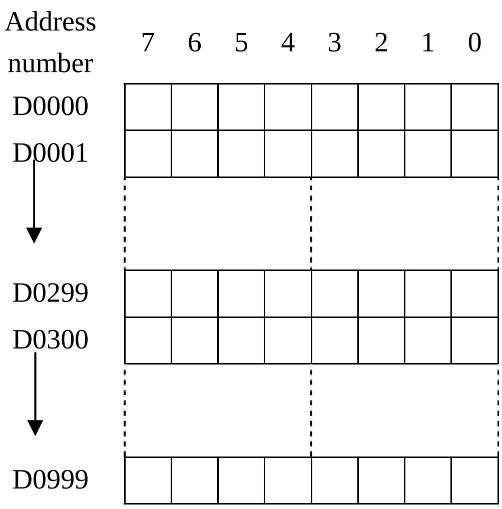
2.11 TIMER PRESET VALUE ADDRESS（DT）

The address area is used for storing preset value of timer and data are saved after the system is switched off. Address range: DT0000～DT0099 and value range: 0～21,4748,3647.



2.12 DATA LIST ADDRESS（D）

D0000～D0299 are zero when CNC is switched on. D0300～D0999 are saved after it is switched off. Value range: 0～255.



2.13 LABEL ADDRESS（L）

It is used for specifying jump target label in JMPB and LBL label.
Range: L0～L9999

2.14 SUBPROGRAM NUMBER（P）

It is used for specifying the target subprogram number to call in CALL and subprogram number in SP.
Range: P0000～P9999

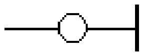
Chapter 3 PLC BASIC INSTRUCTIONS

Basic instructions are used for editing sequential programs and executing 1-bit operation. There are basic instructions for GSK980TD PLC as follows:

Instruction name	Function	Component
LD	Read normally-open contact	X, Y, F, G, R, K, A
LDI	Read normally-closed contact	X, Y, F, G, R, K, A
OUT	Output coil	Y, G, R, K, A
AND	Normally-open contact in series	X, Y, F, G, R, K, A
ANI	Normally-closed contact in series	X, Y, F, G, R, K, A
OR	Parallel normally-open contact	X, Y, F, G, R, K, A
ORI	Parallel normally-closed contact	X, Y, F, G, R, K, A
ORB	Parallel series circuit block	
ANB	Parallel circuit block in series	

3.1 LD，LDI，OUT INSTRUCTION

● Mnemonic code and function

Mnemonic code	Function	Ladder symbol
LD	Read normally-open contact	
LDI	Read normally-closed contact	
OUT	Output coil	

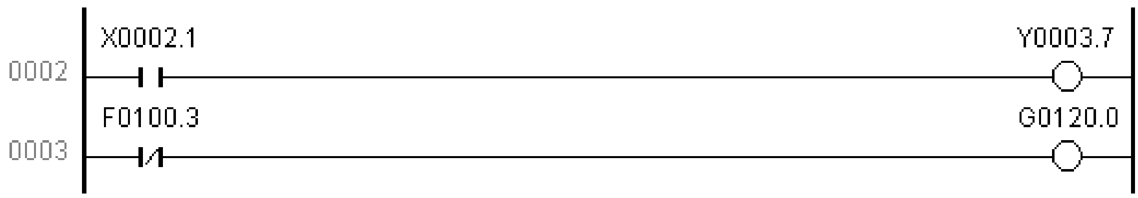
● Instruction explanation

A: LD, LDI are used for connecting contact to bus bar. Each one can combine with instruction ANB and can be used at starting point of branch.

B: OUT is used for driving output relay, internal relay coil instead of input relay.

C: Parallel instruction OUT can be continuously used.

● Programming example



Program explanation：

When X0002.1 is 1, the system outputs Y0003.7

When F0100.3 is 0, the system outputs G0120.0

3.2 AND, ANI INSTRUCTION

- Mnemonic code and function

Mnemonic code	Function	Ladder symbol
AND	Normally-open contact in series	
ANI	Normally-closed contact in series	

- Instruction explanation

AND, ANI can connect one contact in serial. There can be many contacts in serial and the instructions can be used many times.

- Programming example



Program explanation：

Use X0002.1,

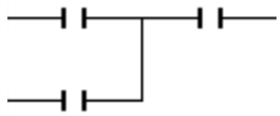
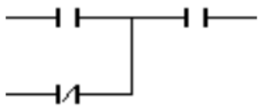
Use F0100.3 and X0002.1 in series

Use X0008.6, and F0100.3 and X0002.1 in series

If X0002.1=1, X0008.6=1 and F0100.3 is 0, the system outputs Y0003.7.

3.3 OR, ORI INSTRUCTION

- Mnemonic code and function

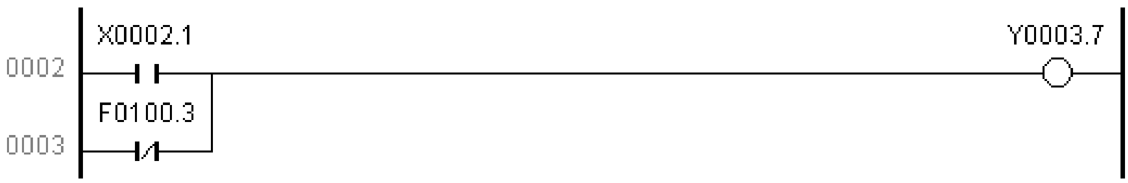
Mnemonic code	Function	Ladder symbol
OR	Parallel normally-open contact	
ORI	Parallel normally-closed contact	

- Instruction explanation

A: OR, ORI can be connected to one contact in parallel. When more than two contacts are connected in series and the serial loop is connected with other loop in parallel, the system should use ORB.

B: The system executes OR, ORI from its current step with LD, LDI in parallel.

- Programming example



Program explanation:

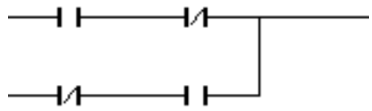
Use X0002.1

Use F0100.3 and X0002.1 in series

If X0002.1=1, and F0100.3 is 0, the system outputs Y0003.7.

3.4 ORB INSTRUCTION

● Mnemonic code and function

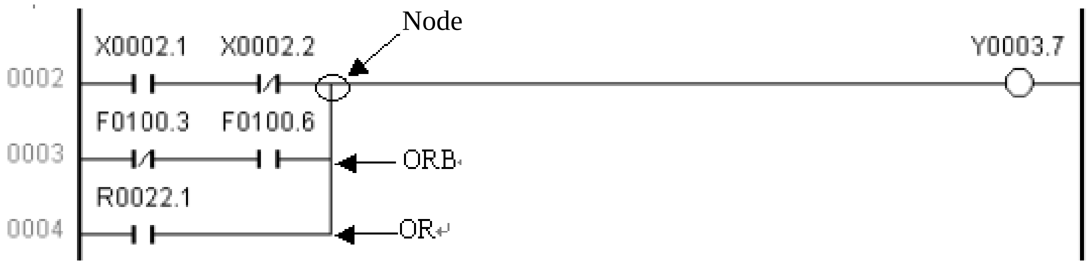
Mnemonic code	Function	Ladder symbol
ORB	Parallel series circuit block	

● Instruction explanation

A: Serial loop block is defined to its loop combined by more than contacts in series. When the serial loop is connected in parallel, starting point of branch uses LD and its end point uses ORB.

B: ORB is sole instruction without address.

● Programming example



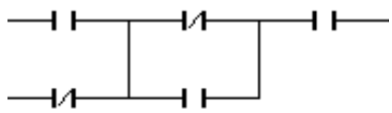
Program explanation:

As above figure, there are three branches(0002,0003,0004) from left bus line to node, and 0002 and 0003 are serial circuit blocks. There is parallel serial circuit block between bus line and node or among nodes, the following ending of branch use ORB except for the first one. Use OR instruction if the branch 0004 is not serial circuit block.

ORB and ANB are instructions without operation function, representing or, and relationship among circuit blocks.

3.5 ANB INSTRUCTION

● Mnemonic code and function

Mnemonic code	Function	Ladder symbol
ANB	Parallel circuit block in series	

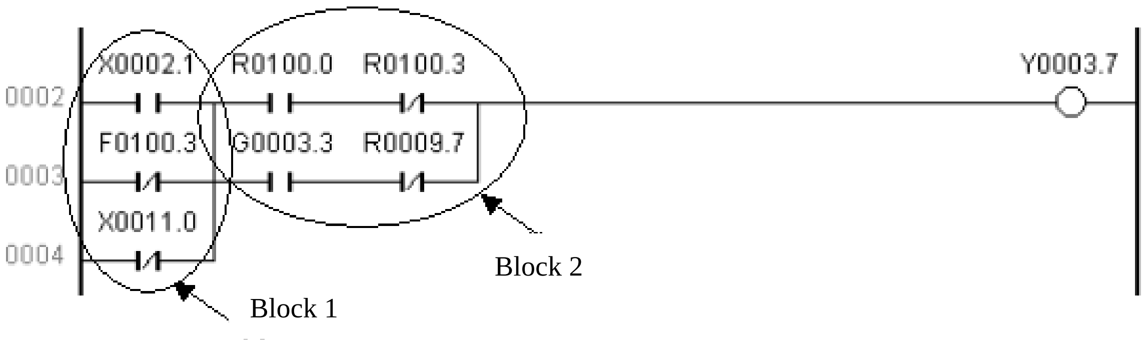
● Instruction explanation

A: Use ANB when the branch loop is serially connected with the previous loop. Use LD, LDI at the starting

point of branch, and use ANB to serially connect with the previous loop.

B: ANB is sole instruction without address.

● Programming example



Program explanation：

As above ladder, ORB represents the parallel serial circuit block in block 2 and ANB represents block 1 and 2 in series.

Chapter 4　PLC FUNCTION INSTRUCTIONS

Some functions are completed by function instructions instead of basic ones. The function instructions in the system are as follows:

Instruction name	Function	Instruction name	Function
END1	End of grade one program	ROTB	Binary rotation control
END2	End of grade two program	DECB	Binary decoding
SET	Set	CODB	Binary code conversion
RST	Reset	JMPB	Program jumping
CMP	Comparative set	LBL	Program jumping label
CTRC	Counter	CALL	Subprogram calling
TMRB	Timer	SP	Start of subprogram
MOVN	Binary Data copy	SPE	End of subprogram
PARI	Parity check	DIFU	Ascending edge set
ADDB	Binary data adding	DIFD	Descending edge set
SUBB	Binary data subtracting	MOVE	And
ALT	Alternative output		

4.1 END1（END of GRADE ONE PROGRAM）

- Instruction function

  There must be END1 in the sequential program one time, located at the end of grade one program. It should be located at the home of the second when there is no grade one program.

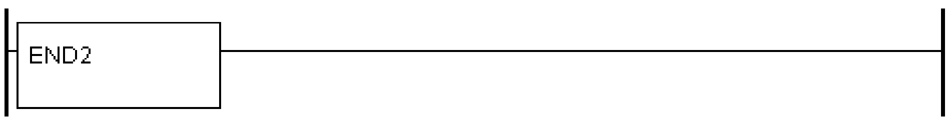
- Ladder format



4.2 END2（END of GRADE TWO PROGRAM）

- Instruction function

  There must be END2 at the end of grade two program which representing end of grade two program.
- Ladder format



4.3 SET

- Instruction function

  Specified address is set to 1.

- Ladder format



- Control condition

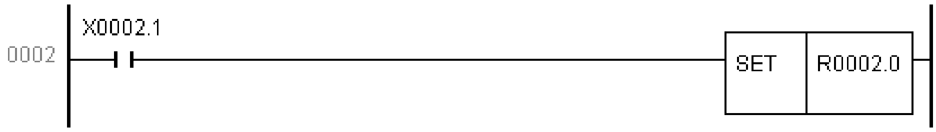
  ACT ＝0：add.b is reserved.

  ＝1：add.b is set to 1.

- Relative parameter

  add.b：set address bit，which can be a contact or output coil, and add is Y, G, R, K or A.

- Program example:



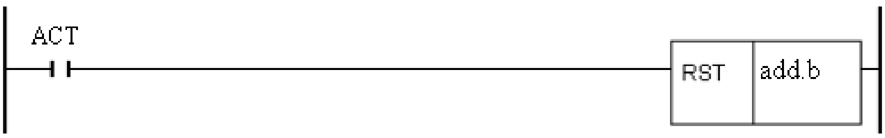
Explanation: When X0002.1 is 1, R0002.0 is set to 1; when X0002.1 is 0, R0002.0 is reserved.

4.4 RST(RESET)

- Instruction function

  Specified address is set to 0.

- Ladder format



- Control condition

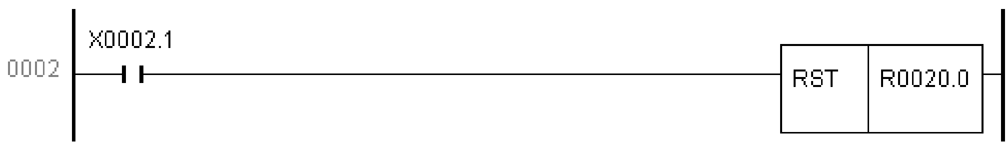
  ACT ＝0:add.b is reserved.

  ＝1:add.b is set to 0.

- Relative parameter

  add.b：reset address bit ,which can a contact or output coil ,and add is Y, G, R, K or A.

- Program example:



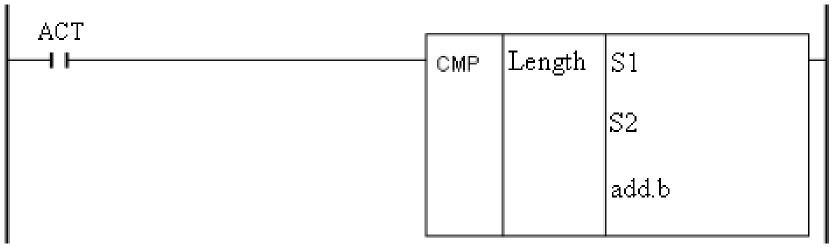
Explanation: When X0002.1 is 0, R0002.0 is reserved; when X0002.1 is 1, R0002.0 is set to 0.

4.5 CMP（BINARY COMPARATIVE SET）

- Instruction function

  Compare two binary data and output its result.

- Ladder format



- Control condition

  ACT ＝0:add.b is reserved.

  ＝1: compare S1, S2 and output the result as follows:

	add.(b+2)	add.(b+1)	add.(b+0)
S1>S2	0	0	1
S1=S2	0	1	0
S1<S2	1	0	0

- Relative parameter

  Length: specify data length, when it is set to 1, 2, 4, the corresponding data length is 1byte, 2bytes,4 bytes.

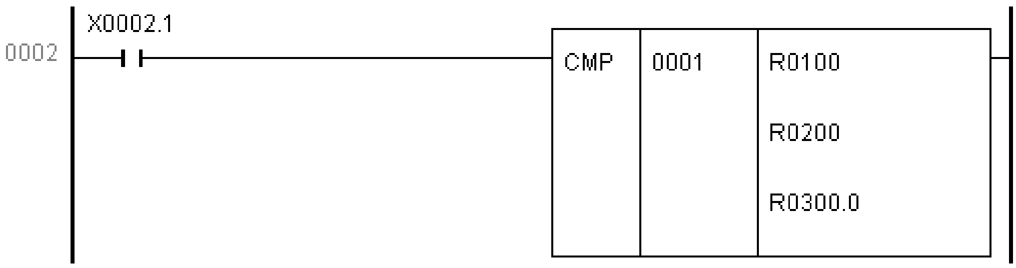
  S1, S2: compare source data 1 with data 2, and the comparative result can be constant or address number(it is not address bit. For example: add.b is illegal.). Address number is R, X, Y, F, G, K, A, D, T, C, DC and DT.

  add.b：it is the comparative result and can be R, Y, G, K and A.

- Relative operation information register:

R900								
			overflow				Negative	Zero

- Program example:



Explanation: When X0002.1 is 0, the system does not compare the data and R0300.0 is reserved;

When X0002.1 is 1, the system compares the data as follows:

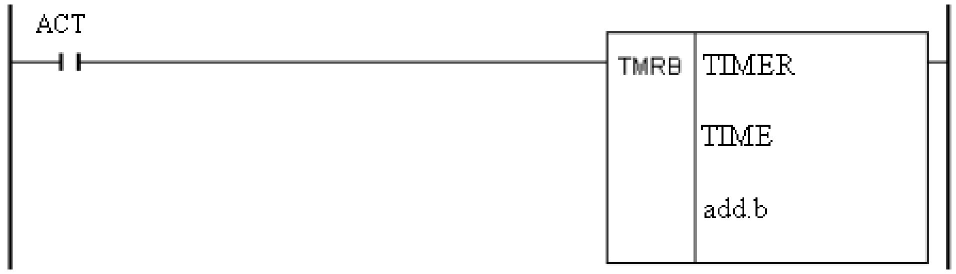
	R0300.2	R0300.1	R0300.0
R0100>R0200	0	0	1
R0100=R0200	0	1	0
R0100<R0200	1	0	0

4.6 TMRB（TIMER）

- Instruction function

  Delay connecting the timer.

- Ladder format

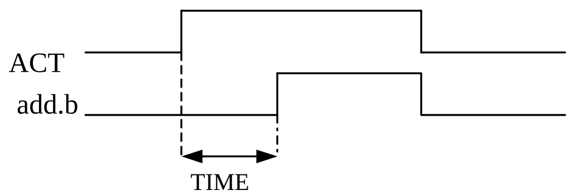


- Control condition

  ACT =0：reset TIMER and add.b.

  =1：TIMER times from 0, and add.b=1 when TIMER reaches the preset time.

  Logic relation is as follows:



- Relative parameter

  TIMER: timer number, range: T0000～T0099.

  TIME: timing constant or data register with DT in front. DT range is from 0 21,4748,3647(ms) .

  add.b：it is timer output address and can be R, Y, G, K and A.

  TIMER is executed every 8ms and timing with 8ms as unit.

- Program example:



Explanation: When X0002.1 is 0, T0002 and R0300.0 are 0;

When X0002.1 is 1, T0002 starts to count, and R0300.0 is set to 1 after it reaches the time set by DT0004.

4.7 CTRC（BINARY COUNTER）

- Instruction function

Data in the counter is binary and the functions of CTRC is as follows:

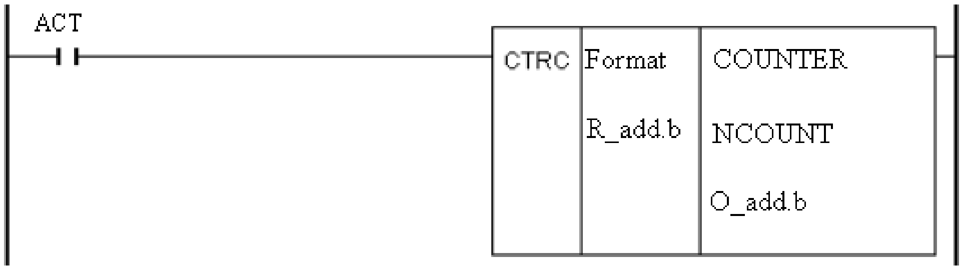
A: reset counter: it resets count value and the system outputs the corresponding signal when the count value reaches the resetting count value.

B: ring counter：input the count value signal when the counter reaches the reset value, and count again the counter reset to initial value.

C: adding/subtracting counter: bit bidirectional counter used for addition and subtraction.

D: initial value selection: it can be 0 or 1.

- Ladder format



- Control condition

ACT is at ascending edge:

Adding: COUNTER counts from its setting initial value, adds the count one time when it reaches the ascending edge, and O_add.b =1 when it reaches the reset count value. O_add.b =0 when COUNTER is less than NCOUNT; COUNTER resumes and starts to count and O_add.b =0 when it reaches the ascending edge again.

Subtracting: COUNTER subtracts from its setting initial value, subtracts the count one time when it reaches the ascending edge, and O_add.b =1 when it reaches the reset count value. O_add.b =0 when COUNTER is more than NCOUNT; COUNTER resumes and starts to count and O_add.b =0 when it reaches the ascending edge again.

ACT＝0：

COUNTER and O_add.b are reserved.

- Relative parameter

Data format:

0	0	CN0	U/D

U/D: Specify adding/subtracting count
0：adding count and counting from 0
1：subtracting count and counting from reset value

CN0: Specify the counter initial value
0：count from 0
1：count from 1

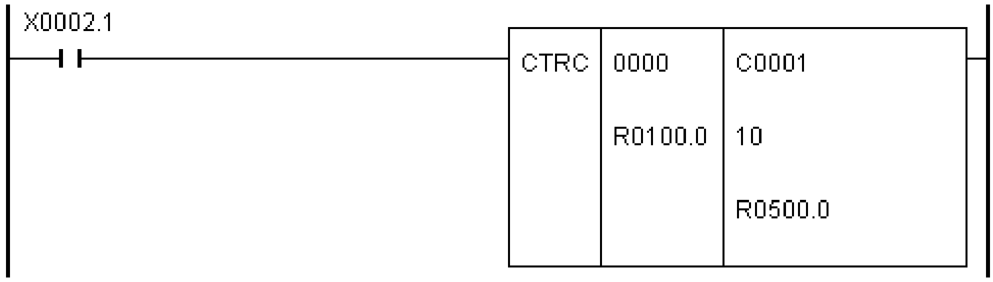
R_add.b ：when it is 1 whatever ACT is , COUNTER＝CN0，O_add.b =0. R_add can be X, Y, G, F, R, K, A.

COUNTER: specify the counter number(Cxxx, xxxx is digit(0～99),

NCOUNT: it is counter preset value(constant) and also can be data register beginning with DC. If it is constant, its value is 0～21,4748,3647.

O_add.b: when it reaches to the count value, the output is 1 and O_add can be R, Y, G, K and A.

- Program example:



Explanation: When R0100.0 is 1, C0001=0 and R0500.0=0；

When R0100.0 is 0, X0002.1 reaches the drift up one time, C0001 adds the count one time, and R0500.0 is 1 if the count reaches 10.

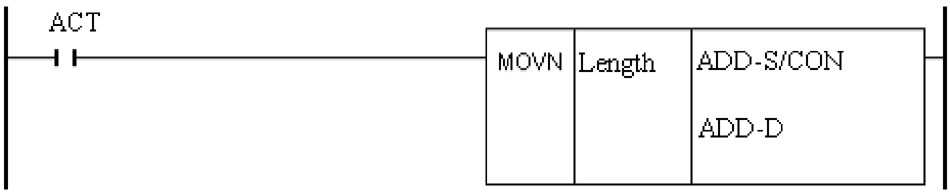
When X0002.1 reaches the drift up again, C0001 is reset to 0 and start to count, and R0500.0 is reset to 0.

4.8 MOVN（BINARY DATA COPY）

- Instruction function

Transmit data at source address or specified binary data to destination address.

- Ladder format



- Control condition

ACT ＝0：ADD-D is reserved.

＝1：copy the value in ADD-S or constant CON to ADD-D

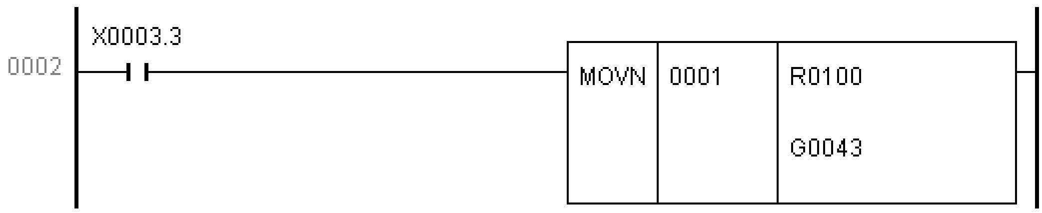
- Relative parameter

Length : it is the copy data(1, 2, 4 byte).

ADD-S/CON: it is initial byte of source data address or constant, and its address number is R, X, Y, F, G, K, A, D, T, C, DC and DT.

ADD-D: it is initial byte of target address, and its address number is R, Y, G, K, A, D, T, C, DC, DT.

- Program example:



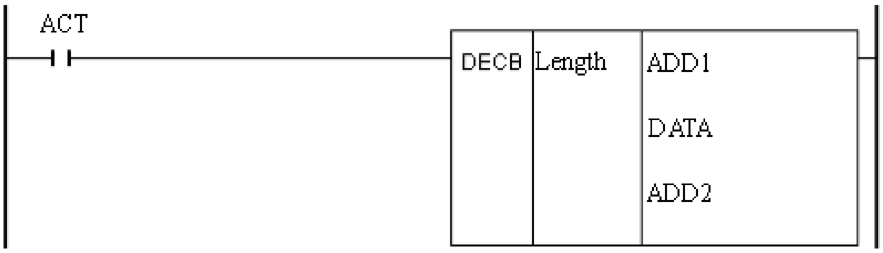
Explanation：R0100 value is transmitted to G0043 when X0003.3 is 1.

4.9 DECB（BINARY ENCODING）

- Instruction function

  DECB can decode the binary code data. When there is the same one between one of 8 successive data and the code data, the corresponding output data is 1; when there is no the same one, the output data is 0. DECB is used for encoding data of M or T function.

- Ladder format



- Control condition

  ACT ＝0：reset the 8 data bits of ADD2. ADD2.

  ＝1：compare the content value of decoding address(ADD1) with 8 successive data beginning with DATA. When the value of ADD1 is equal to one of 8 data and the output address (ADD2)’s corresponding bit which sequence number is that of equal data in these 8 data is set to 1.

- Relative parameter

  Length：specify the length(1, 2, 4) of ADD1.

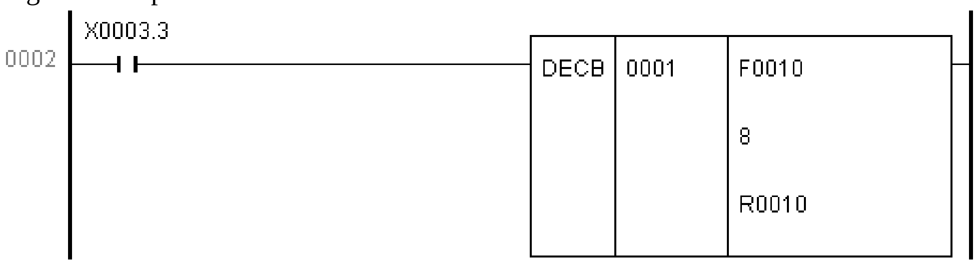
  ADD1: it is initial address of encoding and the address number is R, X, Y, F, G, K, A, D, T, C, DC and DT.

  .

  DATA: reference value of comparative constant.

  ADD2: output comparative result. Its address number is R, Y, G, K and A.

- Program example:



X0003.3=1：

When F0010=8，R0010.0=1；

When F0010=9，R0010.1=1；

…………………………

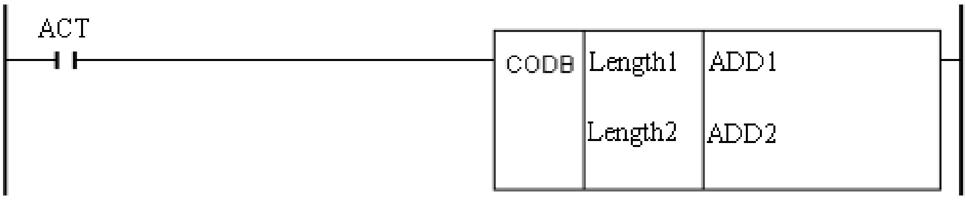
When F0010=15，R0010.7=1

4.10 CODB（BINARY CODE CONVERSION）

- Instruction function

  The instruction is used for binary code conversion.

- Ladder format



- Control condition

  ACT ＝0：ADD2 is reserved.

  ＝1：value of conversing input data address(ADD1) is taken as list number of conversion list, from which the conversion data corresponding to the list number is transmitted to output address(ADD2) of conversion data.

Conversion data list

Converting input data Address ADD1: 001 → Number 001

Number	Value
000	D1
001	D2
002	D3
……..	……….
n	Dn

Value D2 → Converting input data Address ADD2: D2

- Relative parameter

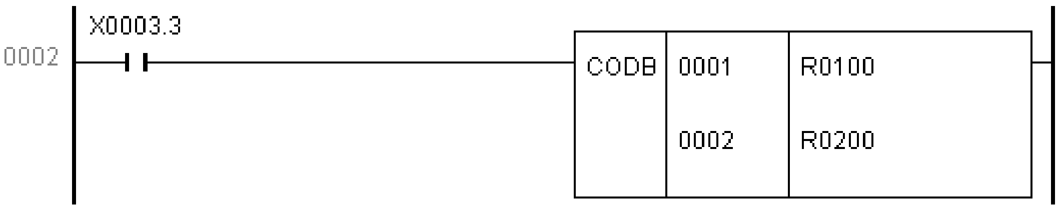
  Length1: binary data length and output address length of conversion data in conversion list, 1-1, 2-2, 4-4 byte.

  Length2：conversion length, 1-2, 2-4, 3-8, 4-16, 5-32, 6-64, 7-128, 8-256.

  ADD1：input address of conversion data. Its address has data with only one byte. Its address is R, X, Y, G, F,A, K, and D.

  ADD2：output address of conversion data and its address is R, Y, G, K and D.

- Program example:



Example

X0003.3=1:

X0003.3＝1, R0100＝0：R0200＝1

X0003.3＝1, R0100＝1：R0200＝2

X0003.3＝1, R0100＝2：R0200＝3

X0003.3＝1, R0100＝3：R0200＝4

Conversion data list

Number	Value
000	1
001	2
002	3
003	4

4.11 JMPB（PROGRAM JUMPING）

- Instruction function

  Immediately jump the program to the position set by label with characteristics as follows: jump instructions can use the same label; forbid jumping END1 and END2; forbid jumping subprogram; jump forward or backward.

- Ladder format



- Control condition

  ACT ＝0：do not jump and execute the next instruction following JMPB.

  ＝1：execute the next instruction following the label after jumping to the specified label.

- Relative parameter

  Lx ： specify the jumping target label, label number must be specified with beginning L address t and it can specify one of L1 to L999.

- Program example:



Explanation: When X0003.3 is 1, the program jumps 0003 and orderly executes from 0004; when X0003.3 is 0, the system orderly executes the program from 0003.

4.12 LBL（PROGRAM JUMPING LABEL）

- Instruction function

  Specify one label in ladder, namely jumping target position specified by JMPB. One Lx is only specified one time by LBL.

- Ladder format



- Relative parameter

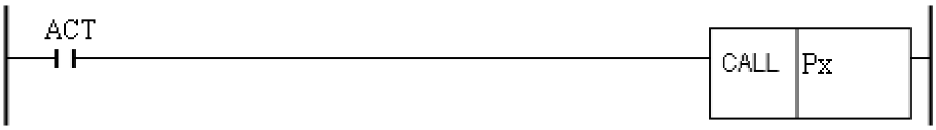
Lx ： specify jumping target label. The label number is specified with beginning L address t and it can specify one of L1 to L999.

4.13 CALL（SUBPROGRAM CALLING）

- Instruction function

Call the specified subprogram with characteristics as follows: call instructions can use the same subprogram; call instruction can be embedded; cannot call a subprogram in grade one program; a subprogram is edited following END2.

- Ladder format



- Control condition

ACT ＝0：execute the next instruction following CALL.

＝1：call subprogram which number is specified.

- Relative parameter

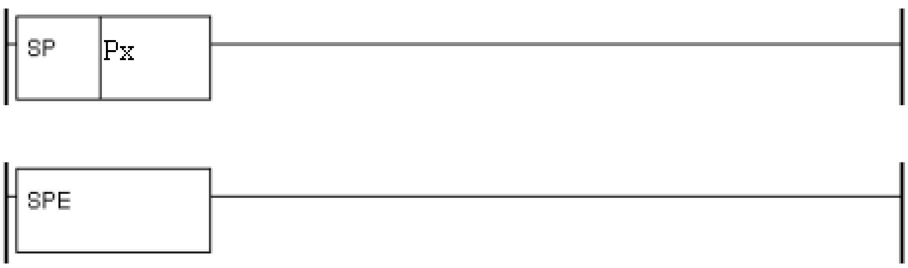
Px ： specify the called subprogram label, its label number is specified with beginning P address and it can specify one of from P1 to P999.

4.14 SP（START of SUBPROGRAM）, SPE（END of SUBPROGRAM）

- Instruction function

SP is to create a subprogram which number is used for subprogram name ,and SPE is taken as end symbol of program ,when SPE is executed, the main program will be returned to. SP and SPE are together used for specifying the subprogram range ,and the subprogram must be edited following END2.

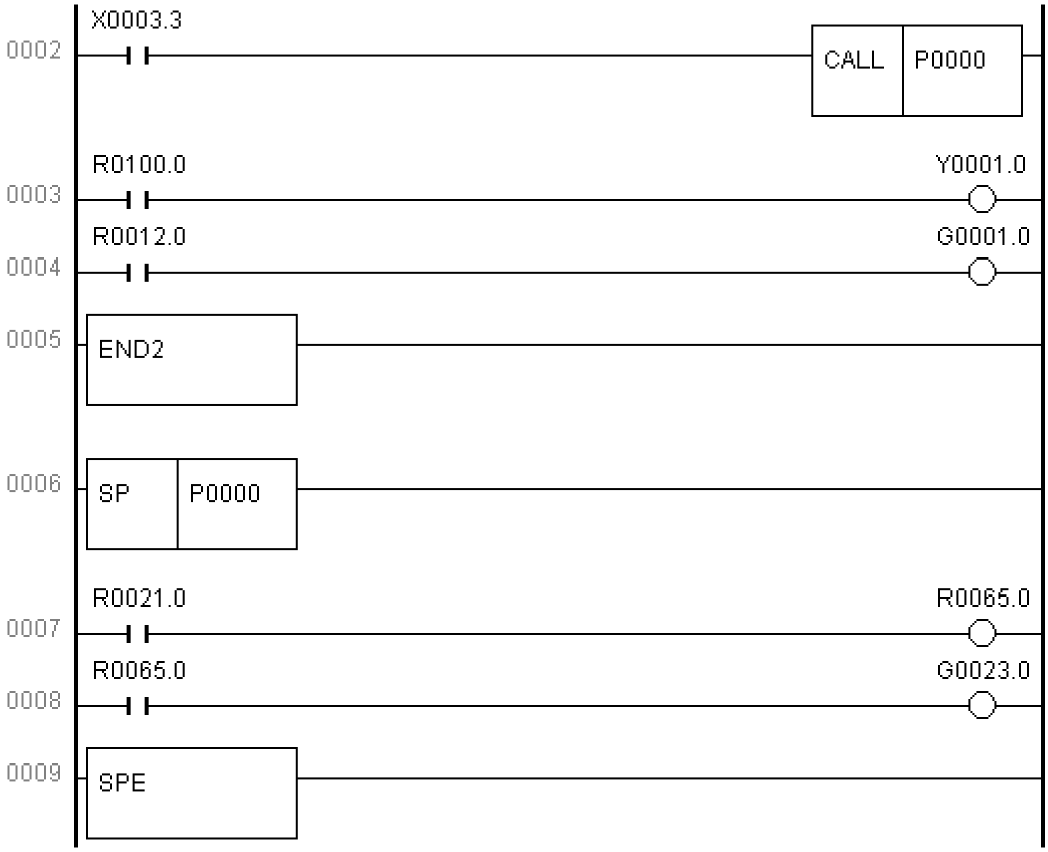
- Ladder format



- Relative parameter

Px ：specify the call subprogram label, its label number is specified with beginning P address and it can specify one of P1 to P999.

- Program example:



Explanation: When X0003.3 is 1, the program calls P0000 subprogram and jumps to execute 0006, and the system orderly executes the program from 0003 after executing 0009.

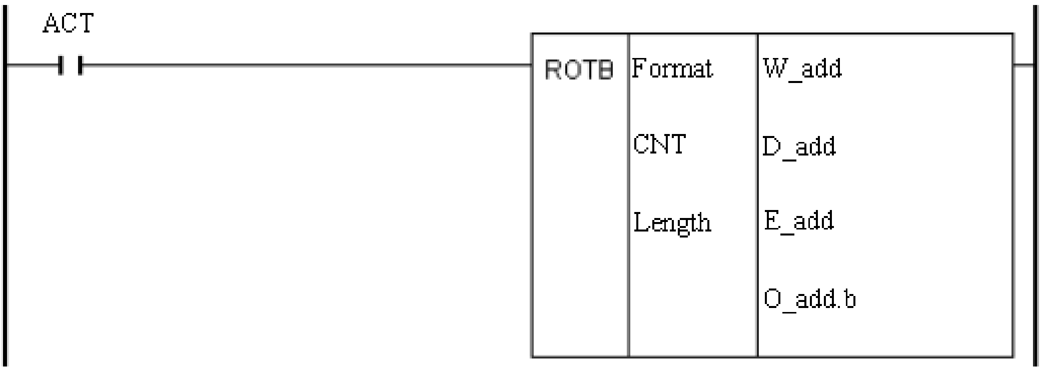
When X0003.3 is 0, the system does not call P0000 subprogram and orderly executes subprograms from 0003.

4.15 ROTB（BINARY ROTATION CONTROL）

- Instruction function

The instruction is used for rotation control, such as toolpost, rotary worktable. Its functions are as follows: to select rotation direction of short path ,to count steps from current position to target position ;or to count steps from the previous one of current position to the previous one of target position ,to count the position number of the previous one of target position.

- Ladder format



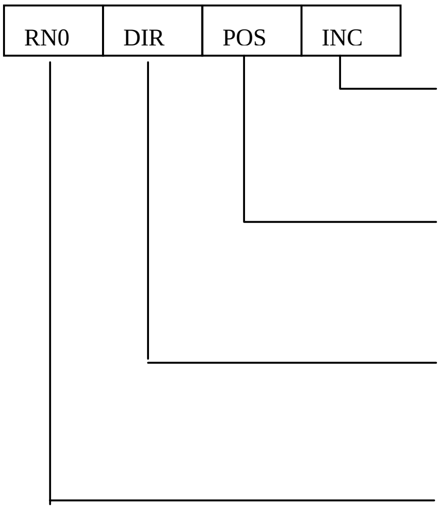
- Control condition

ACT ＝0：do not execute instructions, and E_add and O_add.b are reserved.

＝1：execute instruction and output its result to E_add and O_add.b.

- Relative parameter

Format:



Count position number or steps
0：Count position number
1：Count steps

Count position
0：Count target position
1：Count the position before target

Short circuit selection
0：No selection. Positive rotation，i.e. O_add.b=0
1：Selection. Its direction is defined by specifics

Specify the initial number of swivel table
0：Position number of swivel table is from 0
1：Position number of swivel table is from 1

CNT ： centigrade position number of swivel table.

length ： specify the length of W_add ， D_add and E_add(1, 2, 4 byte).

W_add ： it is current position address used for storing current position number. Its address number is R, X, Y, F, G , K, A, D, DC and DT.

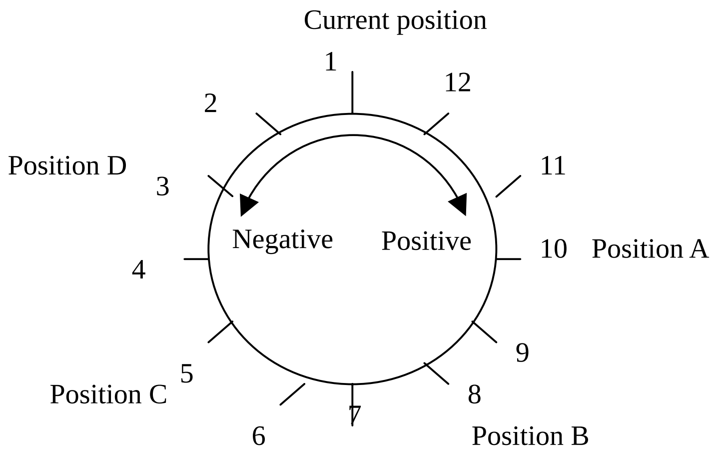
D_add ： it is target position address used for storing target position number. Its address number is R, X, Y, F, G, K, A, D, DC and DT.

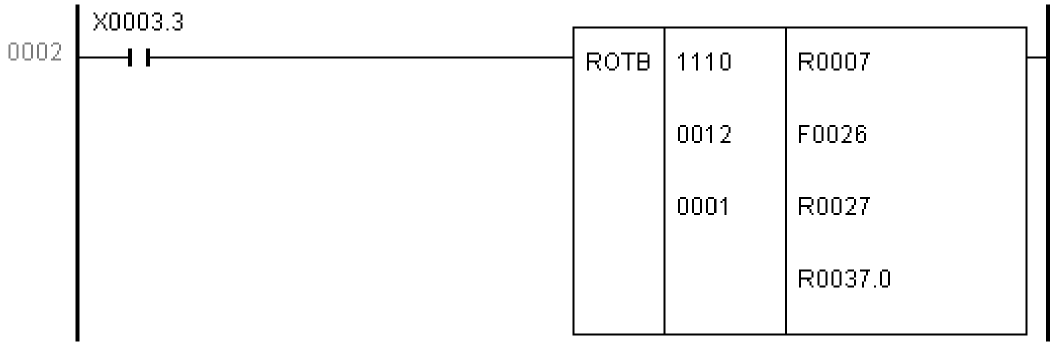
E_add ： it is used for counting the result output address. Its address number is R, Y, G, K, A, D, DC and DT.

O_add.b ： it is used for outputting the rotary direction to ensure adding position number of swivel table is positive(FOR); if it is subtracting, its direction is negative(rev). When O_add.b＝0, the rotary direction is positive; O_add.b＝1, it is negative. Its address number is R, Y, G, K and A.

- Program example:

Example: The current position corresponds to No. 1 tool selection when the toolpost is rotating.





Count the previous position number before target position when the toolpost rotates in short circuit. When the current position number R0007=1 and centigrade position number of swivel table CNT=12, X0003.3=1:

When F0026＝10 and the target position is A, R0027＝11,R0037.0＝1

When F0026＝8 and the target position is B，R0027＝9 ,R0037.0＝1

When F0026＝5 and the target position is C，R0027＝4 ,R0037.0＝0

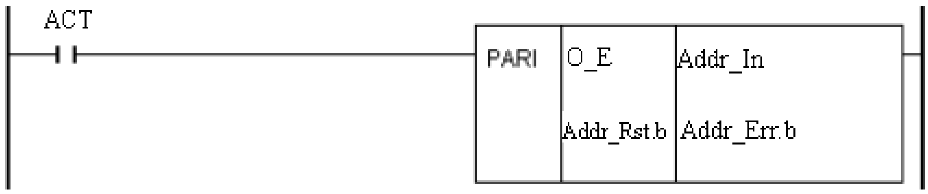
When F0026＝3 and the target position is D，R0027＝2 ,R0037.0＝0

4.16 PARI（PARITY CHECK）

● Instruction function

Check parity of input data ,which is 1 byte(8 bits)

● Ladder format



● Control condition

ACT=1: Execute the parity check for input data. If the input data is inconsistent with the one specified by O_E, Addr_Err.b is 1; otherwise, Addr_Err.b is 0

ACT=0: Do not execute instructions and Addr_Err.b is reserved.

● Relative parameter

O_E =0：number of “1” in put data is even

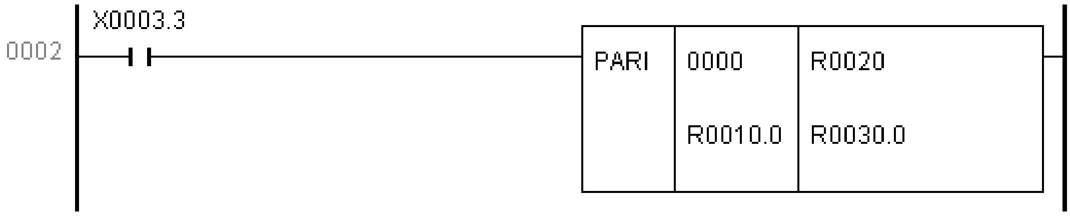
=1：number of “1” in put data is old number

Addr_Rst.b：when it is 1, Addr_Err.b is reset to 0 and its address is X, Y, G, R, F, A, K and K. 为 1 时，Addr_Err.b .

Addr_In ：it is the input data address and its address is X, Y, G, R, F, A, K and D.

Addr_Err.b ：it is the output address of check result and its address is Y, G, R, A and K.

● Program example:

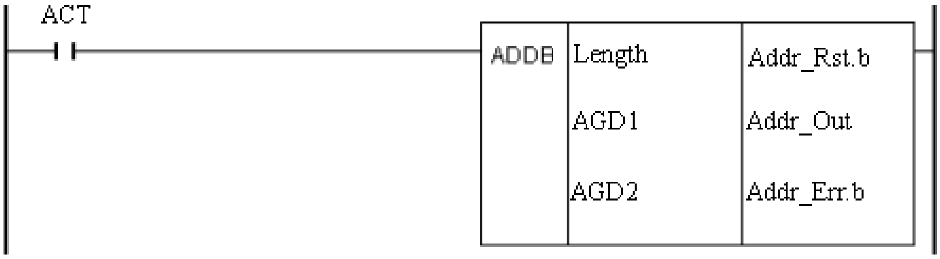


Explanation: When X0003.3 is 1, the system executes PARI instruction, O_E=0000 to execute the even check. When R0010.0 is 1, R0030.0 is reset to 0 and the system does not execute the check. When R0010.0 is 0, it does. When there is an even number for "1"in R0020, R0030.0 is 0. When there is an odd number for "1"in R0020, R0030.0 is 1.

4.17 ADDB（BINARY DATA ADDING）

- Instruction function

  Add binary data
- Ladder format



- Control condition

  ACT=1: the system executes Addr_Out=AGD1+AGD2. If the operation is mistake, Addr_Err.b is 1; otherwise, it is 0.

  ACT=0: the system does not execute instruction, Addr_Out and Addr_Err.b are reserved.

- Relative parameter

  Length ：1-1，2-2，4-4 byte.

  AGD1 ：it is a summand and can be constant or address. Its address number is R, X, Y, F, G, A, K, D, T, C, DC and DT.

  AGD2 ：it is addend and can be constant or address. Its address number is R, X, Y, F, G, A, K, D, T, C, DC and DT.

  Addr_Rst.b：when it is 1, Addr_Err.b is reset to 0 and Addr_Out is reserved. Its address number is R, X, Y, F, G, A and K.

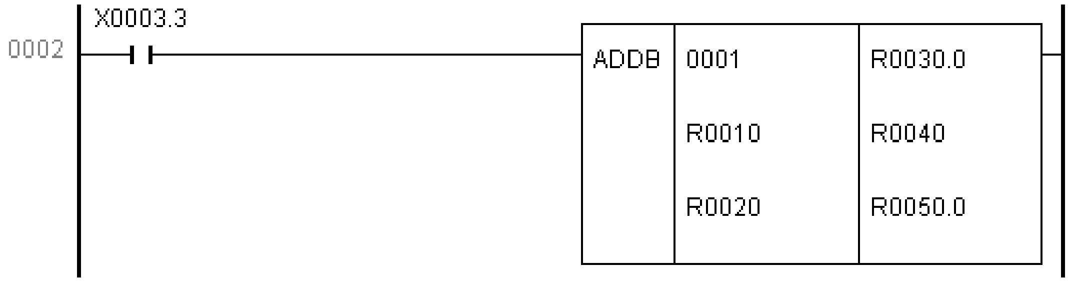
  Addr_Out ：it is the output data address of run result. Its address is Y, G, R, A, K, DC, DT, D, C, and T.

  Addr_Err.b：it is the output address of mistake operation result and its address is Y, G, R, A and K.

- Relative operation information register

R900			**overflow**				**negative**	**zero**

- Program example:



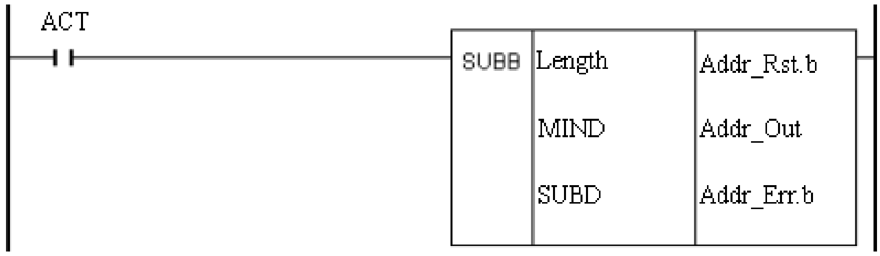
Explanation: When X0003.3=1, the system executes ADDB instruction. When R0040=R0010+R0020 and if the operation is mistake, R0050.0 is 1, otherwise it is 0. When R0030.0 is 1, R0040 is reserved and R0050.0 is reset to 0.

4.18 SUBB（BINARY DATA SUBTRACTING）

- Instruction function

  Subtract the binary data.

- Ladder format



- Control condition

  ACT=1: the system executes Addr_Out= MIND-SUBD. If the operation is mistake, Addr_Err.b is 1; otherwise, it is 0.

  ACT=0: the system does not execute instruction, and Addr_Out and Addr_Err.b are reserved.

- Relative parameter

  Length ： 1-1，2-2，4-4 byte.

  MIND ： it is minend and can be constant or address. Its address number is R, X, Y, F, G, A, K, D, T, C, DC and DT.

  SUBD ： it is subtrahend and can be constant or address. Its address number is R, X, Y, F, G, A, K, D, T, C, DC and DT.

  Addr_Rst.b：when it is 1, Addr_Err.b is reset. Its address number is R, X, Y, F, G, A and K.

  Addr_Out：it is the output data address of run result. Its address is Y, G, R, A, K, DC, DT, D, C, and T.

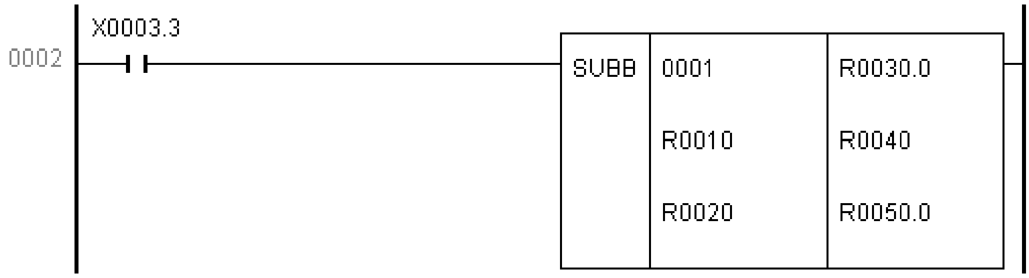
  Addr_Err.b：it is the output address of mistake operation result and its address is Y, G, R, A and K.

- Relative operation information register

R900								
			Overflow				**negative**	**Zero**

Note: the reset has no related to R900.

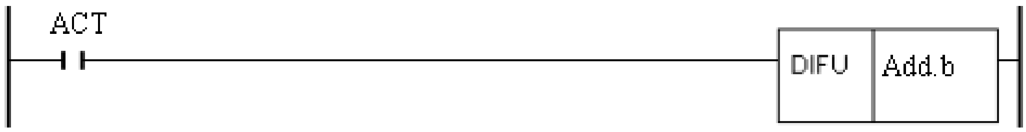
- Program example:



Explanation: When X0003.3=1, the system executes SUBB instruction. When R0040=R0010-R0020 , and if the operation is mistake, R0050.0 is 0, otherwise it is 1. When R0030.0 is 1, R0040 is reserved and R0050.0 is reset to 0.

4.19 DIFU（ASCENDING EDGE SET）

- Instruction function Set the output signal to 1 when the ascending edge of input signal is valid.
- Ladder format



- Control condition

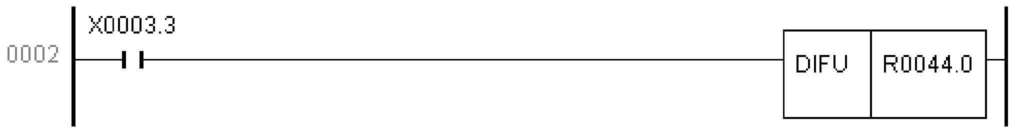
  Input signal ACT: the output signal is set to 1 at the ascending edge of ACT（0—>1）.

  Output signal Add.b: when it is executed, one scanning period of Add.b in ladder is 1.

- Relative parameter

  add.b ： it is the operation result output address and its address is Y, G, R, A and K.

- Program example



Explanation: R0040.0 output is 1 when X0003.3 reaches the ascending edge.

4.20 DIFD（DESCENDING EDGE SET）

- Instruction function

  Set the output signal to 1 when drift down of input signal is valid.

- Ladder format



- Control condition

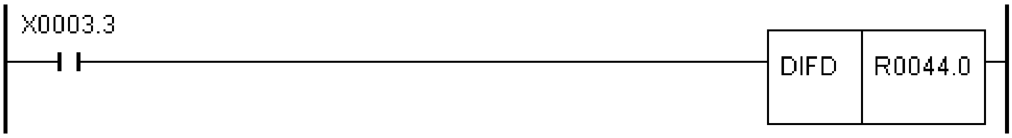
  Input signal ACT: the output signal is set to 1 at the descending edge of ACT（1—>0）.

Output signal Add.b: when it is executed, one scanning period of Add.b in ladder is 1.

- Relative parameter

Add.b ： it is the operation result output address and its address is Y, G, R, A and K.

- Program example:



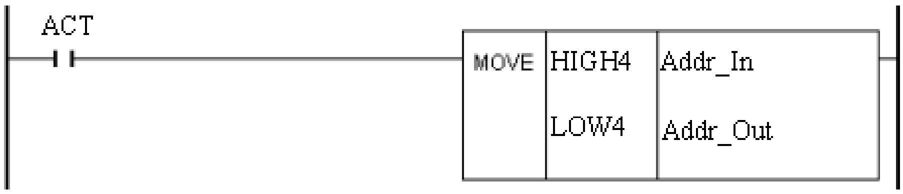
Explanation: R0040.0 output is 1 when X0003.3 reaches the descending edge.

4.21 MOVE（AND）

- Instruction function

Logic multiply and input data execute logic and operation, and then the system output the result to the specified address.

- Ladder format



- Control condition

ACT=1: Logic multiply (HIGH4、LOW4)and input data（Addr_In）execute logical operation, and output the result to the specified address (Addr_Out), which can remove the needless number of bit from 8-bit signal of the specified address.

ACT=0: Addr_Out is reserved.

- Relative parameter

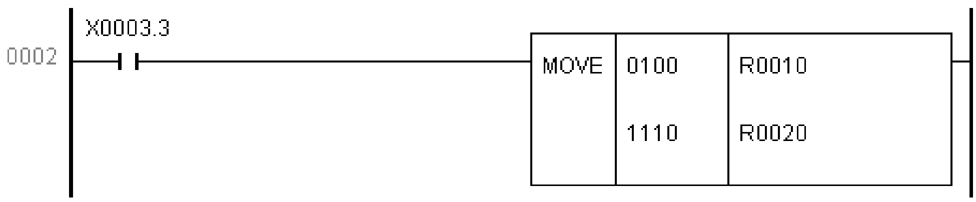
HIGH4 ： high 4-bit logic multiply

LOW4 ： low 4-bit logic multiply

Addr_In ： input data address. Its address number is R, A, K, X, Y, F, G, and D.

Addr_Out： output data address. Its address number is R, A, K, X, Y, F, G, and D.

- Program example:



Explanation: When X0003.3 is 1, logically execute R0010 and 01001110 to save its result to R0020.

4.22 ALT（ALTERNATIVE OUTPUT）

- Instruction function

  It is alternative output instruction. The output signal outputs reversely when the ascending edge of input signal changes(0—>1).

- Ladder format



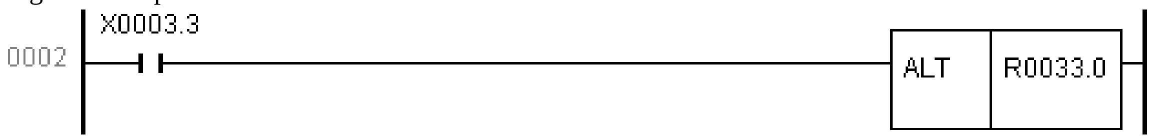
- Control condition

  The output signal Add.b outputs reversely when the input signal ACT changes 0—>1 every time.

- Relative parameter

  Add.b：it is the output signal and its address is Y, G, R, A and K.

- Program example:



  Explanation: R0033.0 oversets one time when X0003.3 reaches the ascending edge every time.

BOOK 2

FUNCTION

Chapter 1　AXES CONTROL SIGNAL

GSK980TD defines 4 axes: Axis 1, 2, 3 and 4. The detailed functions of each axis vary from one software version to another. Now only Axis 1 and 2 are controllable, i.e. Axis X and Z.

1.1 AXIS MOVING STATUS

NC can transmit current axis moving status to PLC, and then PLC works according to the status of axis moving.

1.1.1 Feed Axis and Direction Selection Signal

＋J1～＋J4（G100.0～G100.3）

－J1～－J4（G102.0～G102.3）

● Signal type: PLC→NC

● Signal functions: For detailed functions, please refer to section 3.1.1.

1.1.2 Axis Moving Signal

MV1～MV4（F102.0～F102.3）

● Signal type: NC→PLC

● Signal functions: MV1, MV2, MV3, MV4 are moving signals for Axis 1, 2, 3, 4 respectively. When an axis is moving, NC sets corresponding axis moving signal to 1. When an axis stops moving, the axis moving signal is 0.

● Signal addresses:

F102					**MV4**	**MV3**	**MV2**	**MV1**

1.1.3 Axis Moving Direction Signal

MVD1～MVD4（F106.0~F106.3）

● Signal type: NC→PLC

● Signal functions: MVD1, MVD2, MVD3, MVD4 are axis moving direction signals for Axis 1, 2, 3, 4 respectively. When an axis is moving backwards, NC sets the axis moving direction signal to 1; when an axis is moving forward, the axis moving direction signal is 0; If an axis stops moving, the direction signal will be 1 or 0 depending on the moving status before the axis stopping.

● Signal addresses:

F106					**MVD4**	**MVD3**	**MVD2**	**MVD1**

1.2　SERVO READY SIGNAL

SA（F000.6）

● Signal type：NC→PLC

- Signal functions: When CNC gives out a warning, NC sets SA signal to 0 to stop axis moving. When the warning is cancelled, NC sets SA to 1 to move axis again.
- Signal address:

F0		**SA**						

Chapter 2 PREPARATION for OPERATION

2.1 EMERGENT STOP

Emergency stop signal ESP（X0.5）:

- Signal type: machine tool → NC
- Signal function: If external emergency stop signal X0.5 is not shielded and is detected to be 0, CNC will give out an emergency stop warning (no matter whether PLC program is configured in CNC or not). At this time CNC resets, and the machine tool stops running.
- Signal address:

X0			ESP					

Emergency stop signal ESP（G8.4）:

- Signal type: PLC→NC
- Signal function: If the voltage of G8.4 is 0, once NC detects this signal, an emergency stop warning will be given out.
- Signal address:

G8				ESP				

2.2 CNC READY SIGNAL

MA（F001.7）:

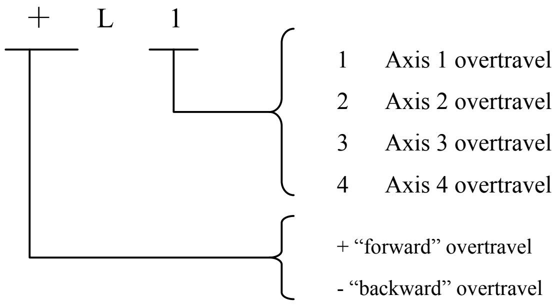
- Signal type: NC→PLC
- Signal function: CNC Ready Signal indicates that CNC is ready for working.
- Output conditions: When CNC is powered on without warning, F1.7 will be set to 1 in several seconds. otherwise, CNC ready signal will be 0.
- Signal address:

F001	MA							

2.3 CNC OVERTRAVEL SIGNAL

+L1~+L4(G114.0~G114.3)
-L1~-L4(G116.0~G116.3)

- Signal type: PLC→NC
- Signal functions: If a tool moves beyond the distance which the machine limit switch sets, the limit switch starts working, the tool slows down and then stops, and overtravel alarm is given out. This signal indicates that control axis already reaches the limit distance. Every direction of each control axis has this signal. "+" and "-" displays the direction, and the number means corresponding axis name.



● Output conditions: If CNC overtravel signal is 0, the control unit will response as follows:

*During automatic operation, if any one of the axis overtravel signal is 0, all axes will slow down and stop, also alarm will be given out.

*During manual operation, only the axis with a moving signal of 0 is going to slow down and stop. The axis can move backwards after it stops.

*Once an axis overtravel signal turns to 0, the moving direction will be saved in the memory. Even though the signal changes to 1 again before the warning is cancelled, the axis cannot move on in the foregoing direction.

● Signal addresses:

G114					**+L4**	**+L3**	**+L2**	**+L1**

G116					**-L4**	**-L3**	**-L2**	**-L1**

2.4 ALARM SIGNAL

AL（F001.0）:

● Signal type: NC→PLC

● Signal functions: When an alarm is given out, the cause message of the alarm will be displayed, and NC will set F1.0 signal to 1. There are two types of alarm: PLC alarm and CNC alarm. When the alarm is cancelled and CNC resets, F1.0 will turns to 0.

● Signal address:

F001								**AL**

2.5 START LOCK SIGNAL

STLK（G007.1）

● Signal type: PLC→NC

● Signal functions: When PLC sets G7.1 to 1, and transmits to NC, NC control axis slows down and stops. During automatic operation, if G7.1 is 1, and the commands before axis moving command block contain M, S, T commands, M, S, T commands can execute constantly until all the axis moving command blocks are finished and axis slows down and stops. Unless G7.1 signal is 0, program can restart.

● Signal address:

G007							**STLK**	

2.6 MODE SELECTION

Mode selection signals include MD1, MD2, MD4, DNC1 and ZRN. Eight types of work mode can be selected: Edit mode, Auto mode, MDI mode, Jog mode, Handwheel mode/Increment mode, Machine Reference Point Return mode and Program Reference Point Return mode. CNC outputs work mode detection signals and informs PLC which working mode CNC is using.

2.6.1 Mode Selection Signal

MD1、MD2、MD4(G043.0～G043.2) DNC1(G043.5) ZRN（G043.7）:

- Signal type: PLC→NC
- Signal function: Work mode codes are as follows.

No.	Code signal / Work mode	ZRN	DNC1	MD4	MD2	MD1
1	Edit mode	0	0	0	1	1
2	Auto mode	0	0	0	0	1
3	MDI mode	0	0	0	0	0
4	Handwheel mode/Increment mode (HANDLE/INC)	0	0	1	0	0
5	Jog mode (JOG)	0	0	1	0	1
9	Machine Reference Point Return mode (REF)	1	0	1	0	1
10	Program Reference Point Return mode (PROG_ZERO)	0	0	0	1	0

PLC sets a value/digit to a Code signal, and then transfers to NC. NC will determine the working method of CNC according to the code signal value.

- Signal addresses:

G43	ZRN		DNCI			MD4	MD2	MD1

2.6.2 Work Mode Detection Signal

MINC(F3.0)、MH(F3.1)、MJ(F3.2)、MMDI(F3.3)、MMEM(F3.5)、MEDT(F3.6)、MREF(F4.5)、MPST(F4.6)

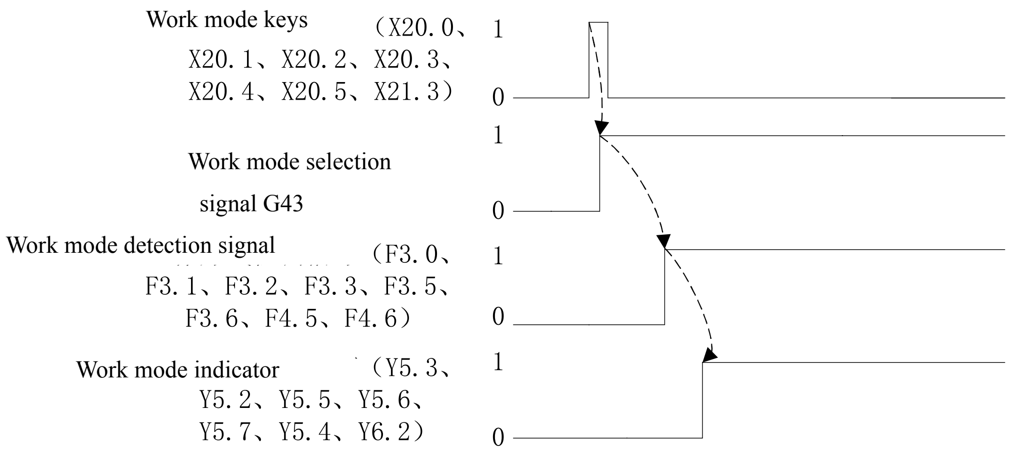
- Signal type: NC→PLC
- Signal functions: when CNC is working under a certain mode, if related "F" signal is set to 1, and transmitted to PLC, PLC can be informed about the current working method.

Increment mode detection signal	MINC
Handwheel mode detection signal	MH
Jog mode detection signal	MJ
MDI mode detection signal	MMDI
Auto mode detection signal	MMEM
Edit mode detection signal	MEDT
Machine Reference Point Return mode detection signal	MREF
Program Reference Point Return mode detection signal	MPRO

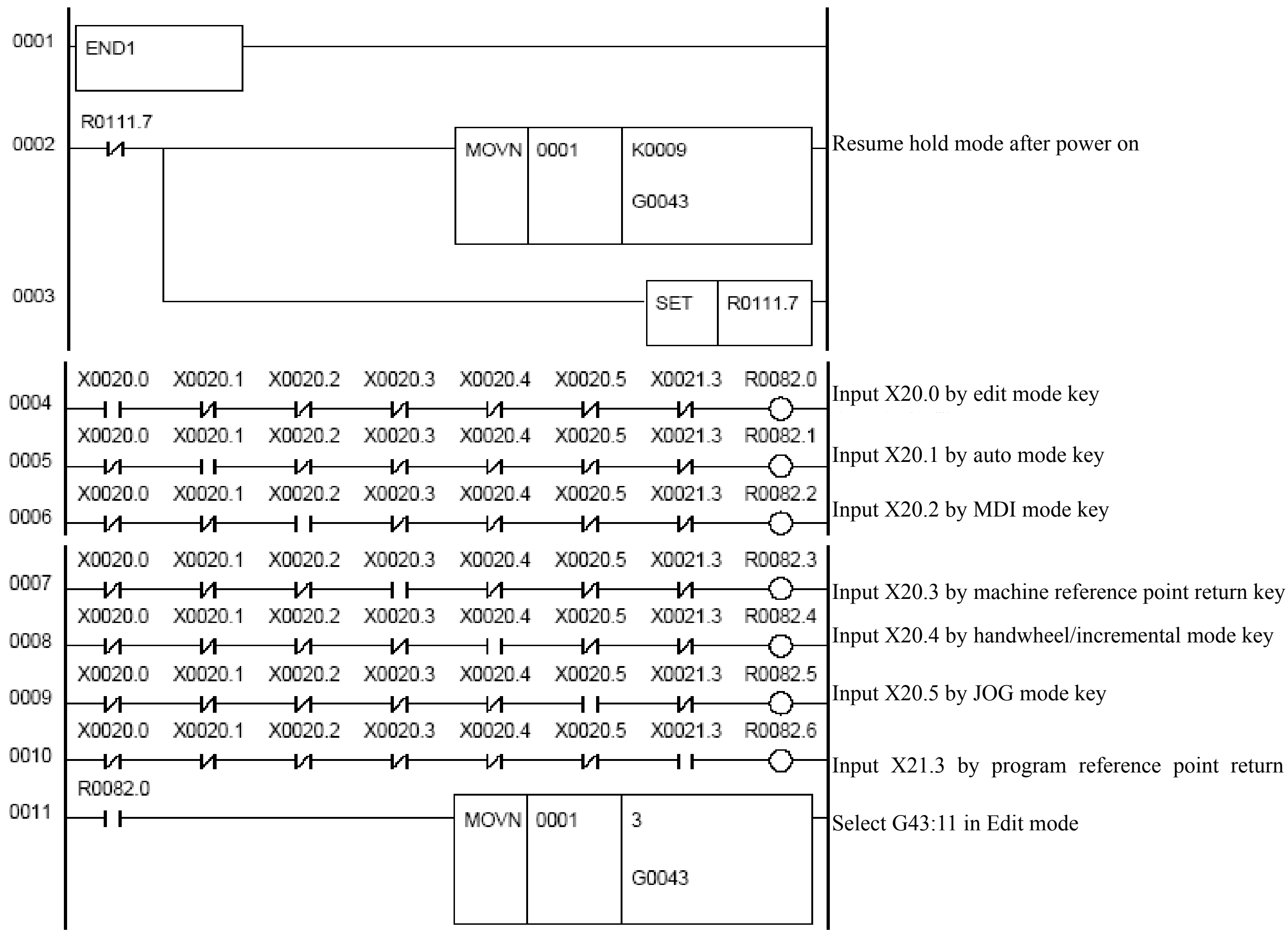
- Signal addresses:

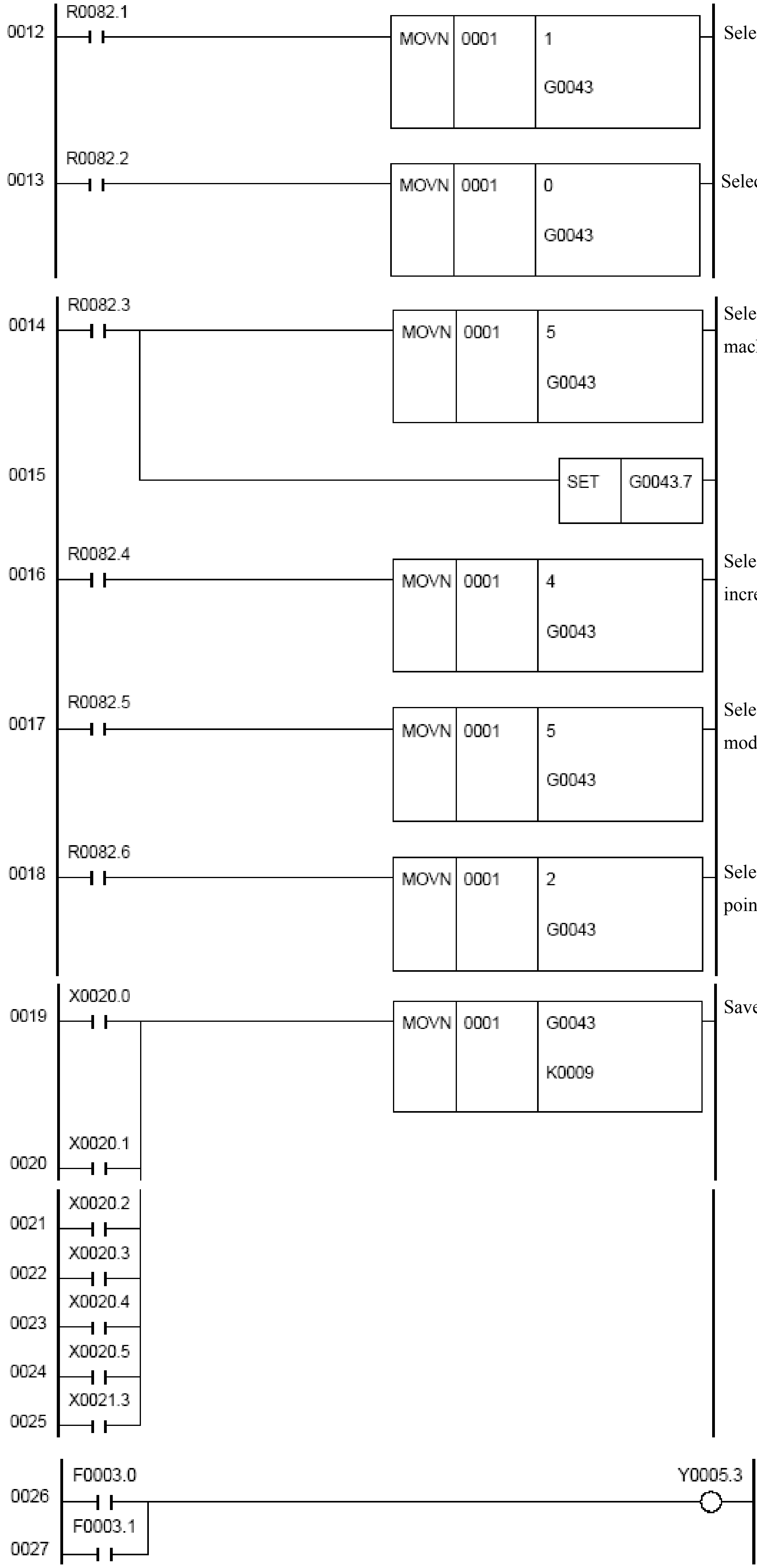
F3		MEDT	MMEM		MMDI	MJ	MH	MINC
F4		MPRO	MREF					

2.6.3 Sequence of Work Mode Signals



2.6.4 Work Mode Control Flowchart





Select G43:1 in Auto mode

Select G43:0 in MDI mode

Select G43:10000101 in machine reference point return mode

Select G43:100 in Handwheel/ incremental mode

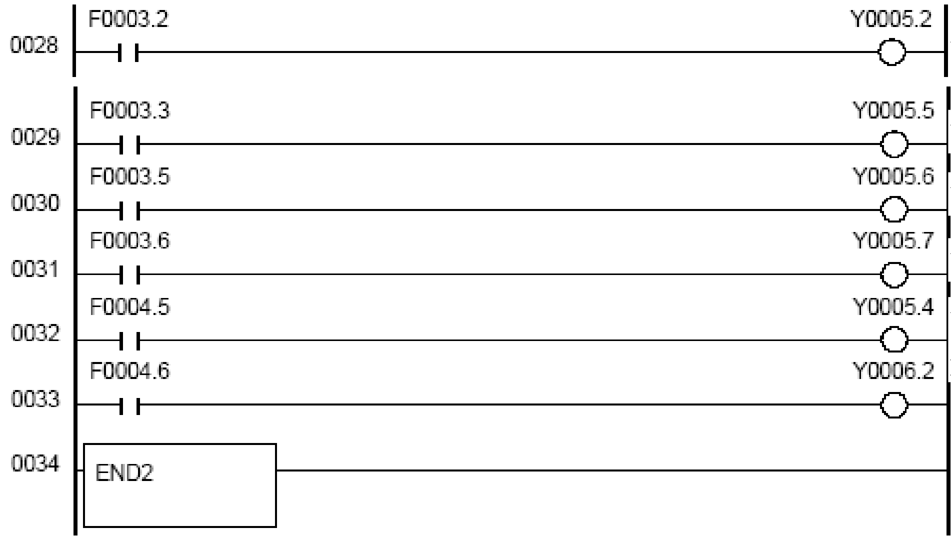
Select G43:101 in Manual mode

Select G43:10 in program reference point return mode

Save G43:100 in current mode

Y5.3: light in Handwheel/ incremental mode

Book 2 Function



Y5.2: light in Manual mode

Y5.5: light in MDI mode

Y5.6: light in Auto mode

Y5.7: light in Edit mode

Y5.4:light in Machine reference point return mode

Y6.2:ligh in program reference point return mode

2.7 STATUS OUTPUT

2.7.1 Rapid Feed Signal

RPDO（F002.1）:

- Signal type: NC→PLC
- Signal function: CNC feeds signals rapidly
- Notes: When an axis feeds rapidly, NC sets F2.1 to be 1. If the feeding stops, F2.1 remains at the same status, and then selects the “un-rapid” feeding, moves the axis or resets CNC, then F2.1 will change to 0.
- Signal address:

F2							**PRDO**	

2.7.2 Cutting Feed Signal

CUT（F002.6）:

- Signal type: NC→PLC
- Signal function: when CNC is performing cutting feed (linear interpolation, circular interpolation, helical interpolation, thread cutting, skip cutting or fixed cycle cutting), NC sets F2.6 to 1, and then transmits to PLC. When it changes to rapid feed, F2.6 turns to be 0.
- Signal address:

F2		**CUT**						

2.7.3 Tap Cutting Signal

TAP（F1.5）:

- Signal type: NC→PLC
- Signal functions: when CNC is performing tap cutting, NC sets F1.5 to 1.When CNC is not performing tap cutting, and under the state of emergency stop or reset, NC sets F1.5 to 0.
- Signal address:

F1			**TAP**					

2.7.4 Thread Cutting Signal

THRD（F2.3）：

- Signal type: NC→PLC
- Signal functions: when CNC is executing thread cutting, NC sets F2.3 to 1. While on the contrary F2.3 is 0.
- Signal address:

F2					THRD			

Chapter 3　MANUAL OPERATION

3.1 JOG FEED/INCREMENTAL FEED

Jog feed: In the jog mode, if the feed axis and direction selection signal is set to 1, the tool moves continuously along the selected axis in the selected direction.

Incremental feed: In the incremental mode, if the feed axis and direction selection signal is set to 1, the tool moves one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step can be 0.001, 0.010 or 0.100 times .

The only difference between jog feed and incremental feed is the method of selecting feed distance. In the jog mode, when the feed axis and direction selection signal ＋J1、－J1、＋J2、－J2、＋J3、－J3 is 1, the tool can move/feed continuously. While in the incremental mode, the tool only moves one step.

By pressing the jog rapid traverse feed selection switch, the tool can be moved at the rapid traverse federate. In the incremental feed mode, the distance of each step can be selected by MP1, MP2.

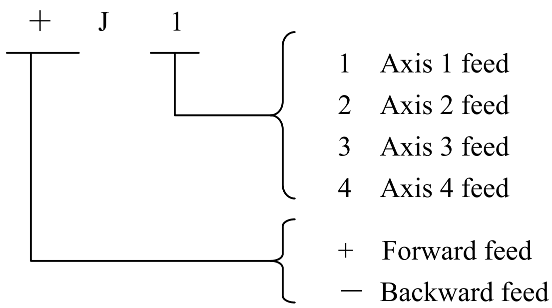
3.1.1 Feed Axis and Direction Selection Signal

＋J1～＋J4（G100.0～G100.3）

－J1～－J4（G102.0～G102.3）

● Signal type: PLC→NC

● Signal functions: In the jog feed or incremental feed mode, select a feed axis and a direction, perform axis moving, and NC sets the corresponding axis and direction selection signal to 1. "＋", "－" indicates feed direction. The number indicates axis number.



● Notes:　A：In the jog feed mode, CNC makes the selected axis moving continuously. In the incremental feed mode, CNC makes the selected axis moving according to the specified rate defined by MP1, MP2 signals.

B：When an axis is moving, NC sets the axis and direction selection signal to 1. When the axis stops moving, the signal changes to 0.

● Signal addresses:

G100					**+J4**	**+J3**	**+J2**	**+J1**
G102					**-J4**	**-J3**	**-J2**	**-J1**

3.1.2 Manual Feed Override Signal

JV00～JV15（G10、G11）：

- Signal type: PLC →NC
- Signal functions: Select manual feed override. The following table shows the relationship between signals and manual feed override. PLC sets a value to G10, G11 and transmits to NC. Then NC sets override based on the value of G10, G11.

G11	G10	Override(%)	Feed speed (mm/min)
0000 0000	0000 1111	0	0
0000 0000	0000 1110	10	2.0
0000 0000	0000 1101	20	3.2
0000 0000	0000 1100	30	5.0
0000 0000	0000 1011	40	7.9
0000 0000	0000 1010	50	12.6
0000 0000	0000 1001	60	20
0000 0000	0000 1000	70	32
0000 0000	0000 0111	80	50
0000 0000	0000 0110	90	79
0000 0000	0000 0101	100	126
0000 0000	0000 0100	110	200
0000 0000	0000 0011	120	320
0000 0000	0000 0010	130	500
0000 0000	0000 0001	140	790
0000 0000	0000 0000	150	1260

- Signal addresses:

G10	**JV07**	**JV06**	**JV05**	**JV04**	**JV03**	**JV02**	**JV01**	**JV00**
G11	**JV15**	**JV14**	**JV13**	**JV12**	**JV11**	**JV10**	**JV09**	**JV08**

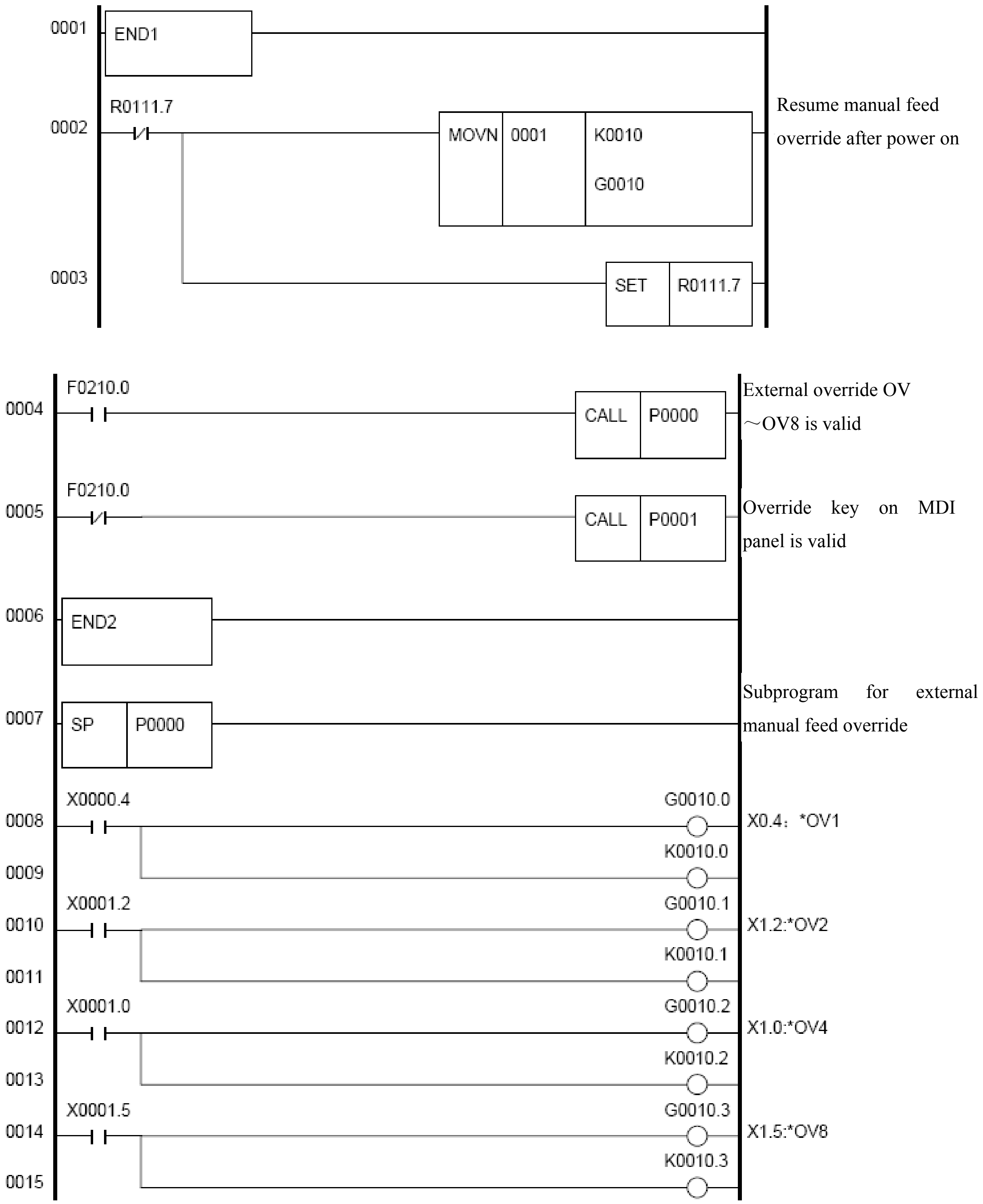
3.1.3 Rapid Traverse Selection Signal

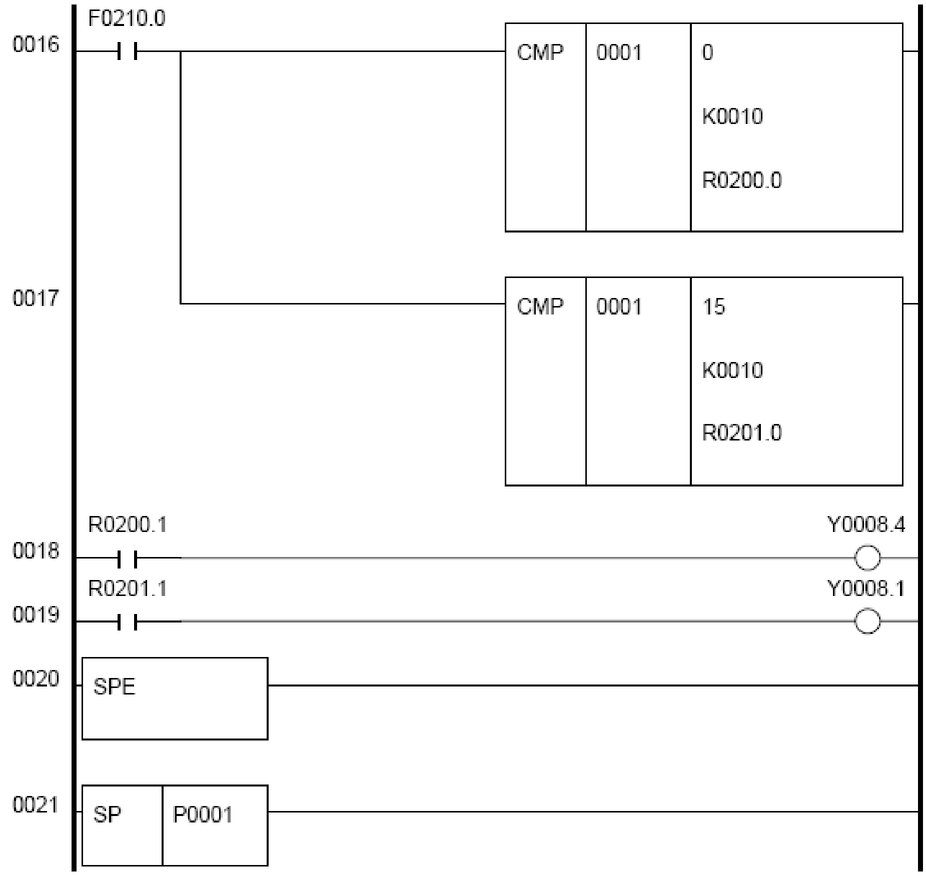
RT（G019.7）

- Signal type: PLC→NC
- Signal functions: PLC sets G19.7 to 1 and transmits to NC to select jog rapid traverse. During the time of rapid traverse, either G19.7 changes from 1 to 0, or from 0 to 1, feed speed will drop to 0 at first, and then increase up to stated value. In the process of speed decreasing, feed axis and direction selection signal remains unchanged.
- Signal addresses:

G19	**RT**							

3.1.4 Manual Feed Override Control Flowchart





Min.: override

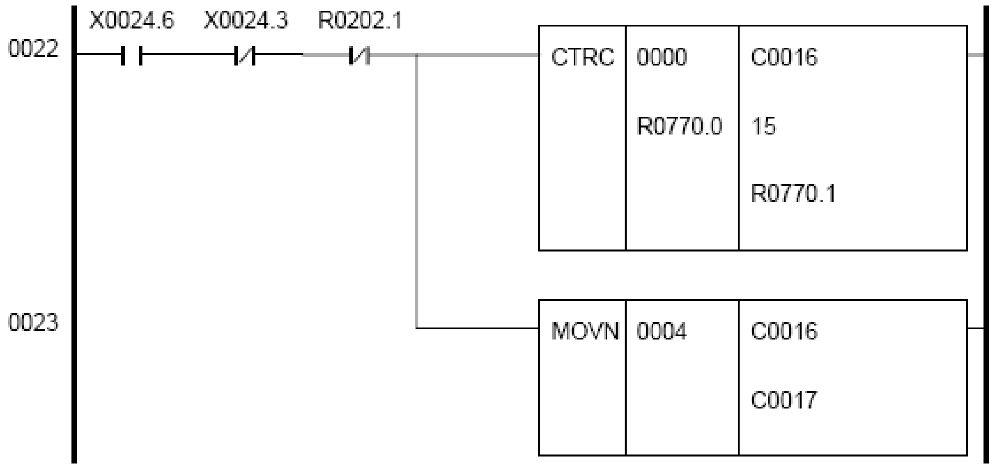
Max. override

Y8.4: light for min. override in manual feed

Y8.4: light for min. override in manual feed

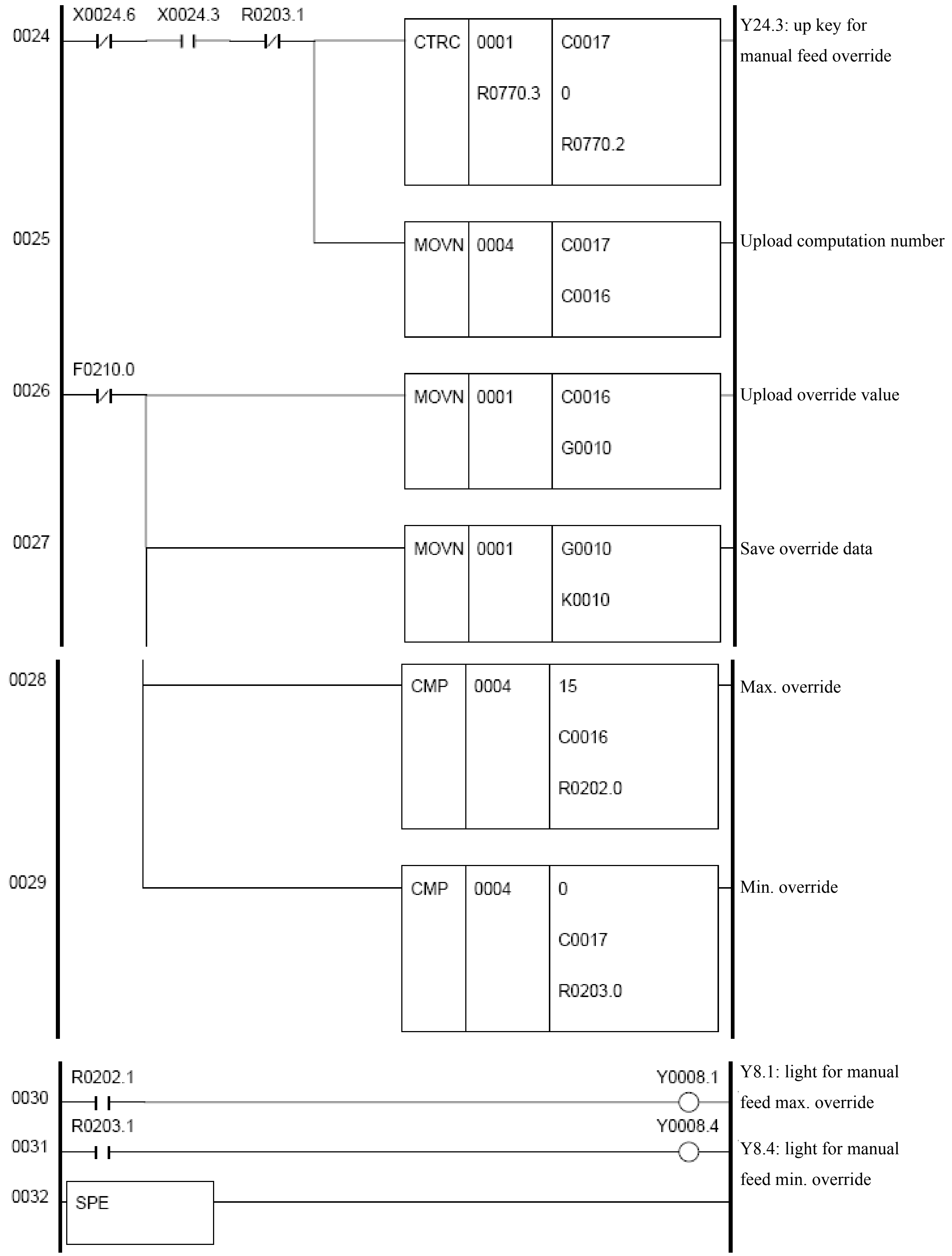
Y8.1: light for max. override in manual feed

Subprogram for manual feed override on MDI panel



X24.6: down key for manual feed override

Upload computation number



3.2 HANDWHEEL FEED

In manual handle feed mode, the tool can be minutely moved by rotating the handle. An axis can be selected according to the handle feed axis selection signal.

3.2.1 Handwheel Feed Axis Selection Signal

HS1A（G18.0）、HS1B（G18.1）

● Signal type: PLC→NC

● Signal type: PLC sets a value to handwheel feed axis selection signal G18.0, G18.1, and transmits to NC. NC selects appropriate axis to feed based on the value of G18.0、G18.1. The relationship between signals G18.0, G18.1 and handwheel feed axis is as follows:

HS1B（G18.1）	**HS1A（G18.0）**	**Feed axis**
0	0	No feed axis selected
0	1	Axis 1 is selected
1	0	Axis 2 is selected
1	1	Axis 3 is selected

● Signal addresses:

G18							**HS1B**	**HS1A**

3.2.2 Handwheel / Increment Override Signal

MP1(G19.4)、MP2(G19.5):

● Signal type: PLC→NC

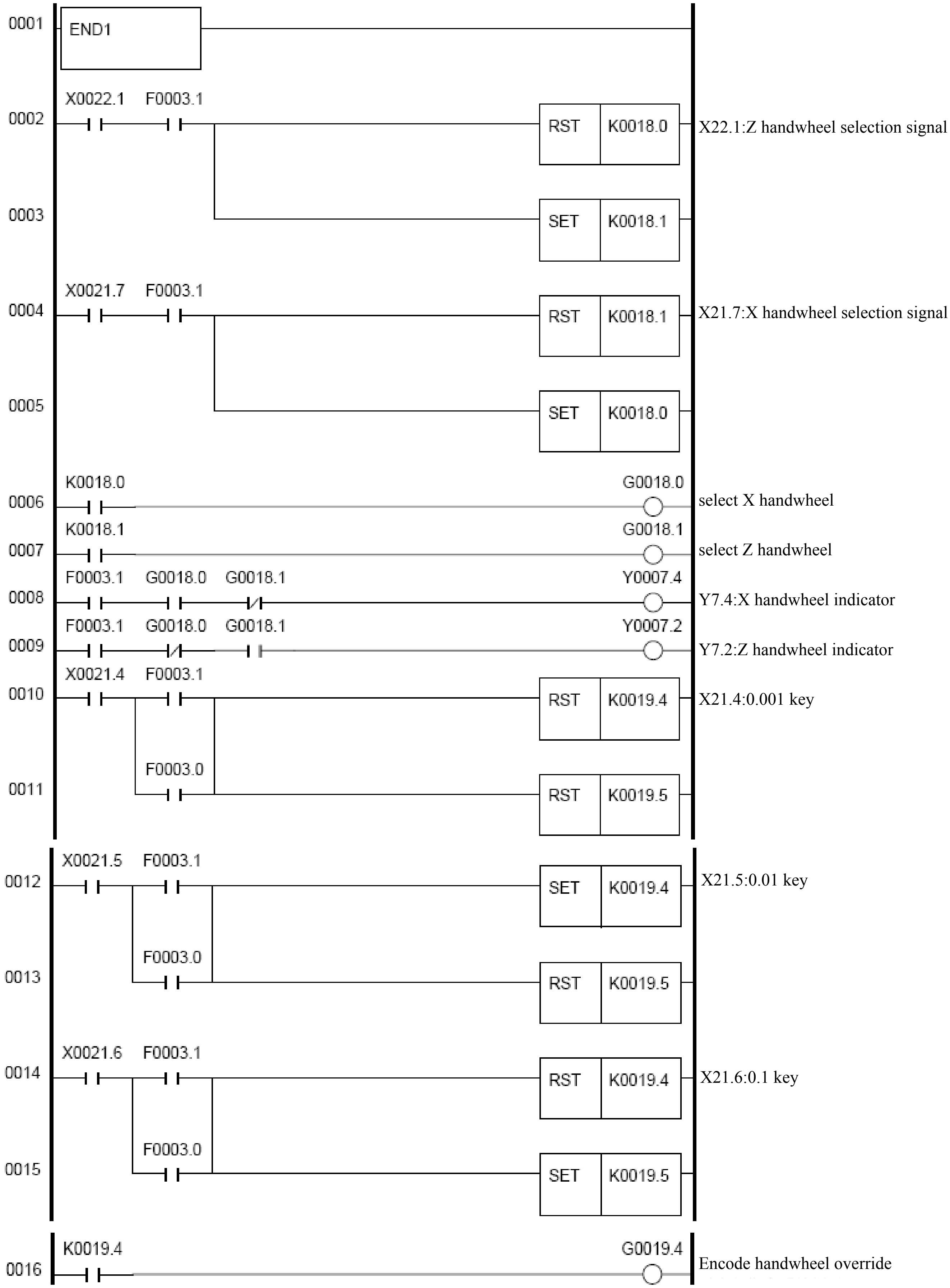
● Signal functions: MP1, MP2 can select the moving distance of each pulse during the handwheel feed, and also can select the moving distance of each step during the incremental feed. PLC sets a value to G19.4, G19.5, and transmits to NC. NC adjusts override according to the value of G19.4, G19.5. See below table for detailed relationship:

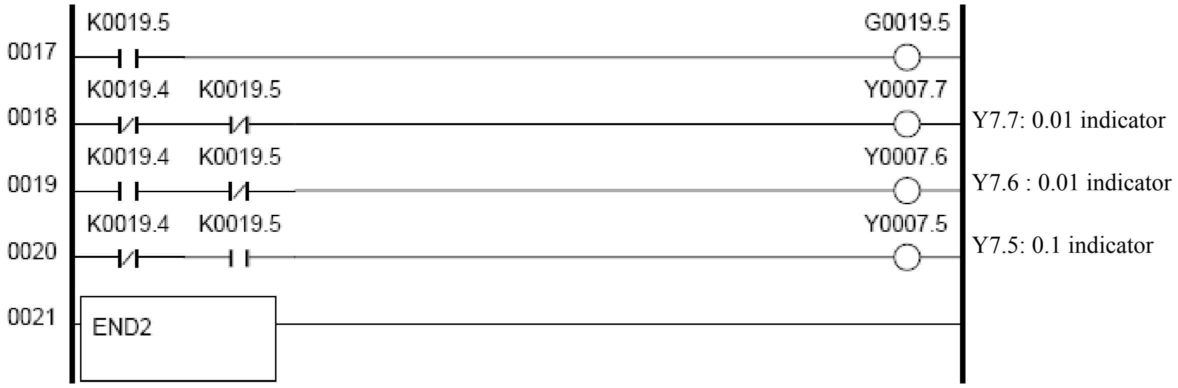
MP2（G19.5）	**MP1（G19.4）**	**Moving distance (mm)**
0	0	0.001
0	1	0.010
1	0	0.100
1	1	1.000

● Signal addresses:

G19							**MP2**	**MP1**

3.2.3 Handwheel Override Control Flowchart




0017
K0019.5
G0019.5
0018
K0019.4
K0019.5
Y0007.7
Y7.7: 0.01 indicator
0019
K0019.4
K0019.5
Y0007.6
Y7.6 : 0.01 indicator
0020
K0019.4
K0019.5
Y0007.5
Y7.5: 0.1 indicator
0021
END2

Chapter 4　MACHINE／PROGRAM REFERENCE POINT RETURN

4.1 MACHINE REFERENCE POINT RETURN

In the mode of Machine reference point Return, by setting the feed axis and direction selection signals to 1, the tool can move along the specified direction, and return to machine reference point position. After returning to machine reference point position, CNC sets a workpiece coordinate system based on the values of NO.049, NO.050.

4.1.1 Machine Reference Point Return End Signal

ZP1～ZP4(F094.0～F094.3):

● Signal type: NC→PLC

● Signal function: When an axis ends the machine reference point return, NC will set the related F signal to 1, and transmit to PLC. This is to inform the axis that the machine reference point return already ends. ZP1, ZP2, ZP3, ZP4 signals are machine reference point return end signals for Axis 1, 2, 3, 4 respectively.

● Notes: When machine reference point return has already ended (and G28 command returns to zero), and the current position is within the specified area, the machine reference point return end signal turns to 1. When the tool moves away from machine reference point point, the machine reference point return end signal changes to 0.

4.1.2 Deceleration Signal in Machine Reference Point Return

XDEC（X09.0）、ZDEC（X09.1）:

● Signal type: machine tool→NC

● Signal functions: As soon as NC receives the deceleration signal in machine reference point return, the moving speed will be decelerated, until the machine tool returns to machine reference point position at a low rate. XDEC, ZDEC signals are machine reference point return deceleration signals for Axis X, Z respectively.

● Signal addresses:

X9							**ZDEC**	**XDEC**

4.1.3 Machine Reference Point Setting Signal

ZRF1～ZRF4(F120.0～F120.4)

● Signal type: NC→PLC

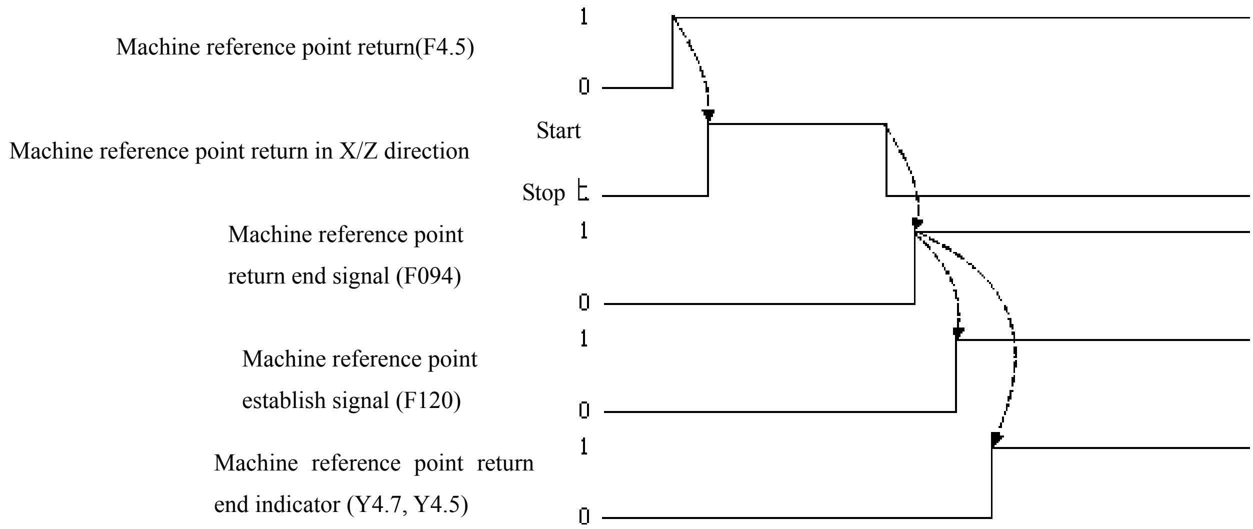
● Signal functions: After ending NC machine reference point return and setting a machine reference point point, the corresponding machine reference point establish signal is set to 1, and then it is transmitted to PLC. ZRF1, ZRF2, ZRF3, ZRF4 signals are machine reference point establish signals for Axis 1, 2, 3, 4 respectively.

Book 2 Function

● Signal addresses:

F120								
					ZRF4	ZRF3	ZRF2	ZRF1

4.1.4 Sequence of Machine Reference Point Return Signal



4.2 PROGRAM REFERENCE POINT RETURN

Under the status of Program reference point Return, Program reference point Return executes and makes the tool moving to G50 position at a rapid speed. While during the time of program automatic executing, if a program stops, the program returns to zero, the tool will return to program start position, and cancel tool offset. If program reference point is not saved in the memory, when the program reference point return command is executing, an alarm occurs. If program reference point is saved, then the program is started up, the position of the tool when G50 block is executing will be saved in the memory automatically. Once the zero point is saved, it will keep unchanged until a new zero point is saved. i.e. when executing Program A, zero point A is saved; Then when executing Program B (if G50 is included), zero point A will be replaced by zero point B.

4.2.1 Program Reference Point Return End Signal

PRO1～PRO4（F198.0～F198.3）:

● Signal type: NC→PLC

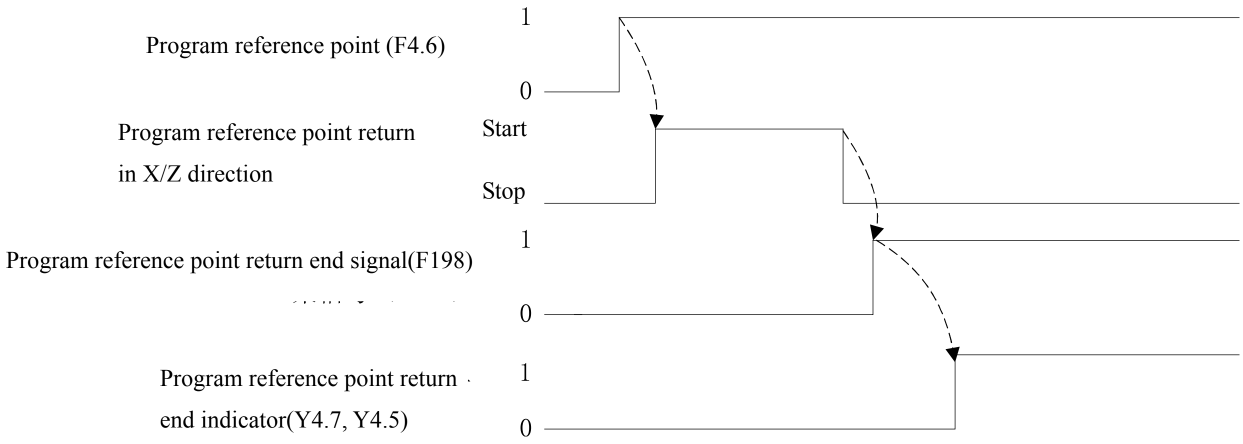
● Signal functions: When program reference point return ends, the tool will stay at the program reference point point. NC will set the program reference point return end signal to 1, and transmit to PLC, which informs PLC that the program reference point return is ended. PRO1, PRO2, PRO3, PRO4 are program reference point return end signals for Axis 1, 2, 3, 4 respectively.

● Note: When the tool moves away from program reference point, the signal of program reference point return turns to 0.

● Signal addresses:

F198								
					PRO4	PRO3	PRO2	PRO1

4.2.2 Sequence of Program Reference Point Return Signals



Chapter 5 AUTOMATIC OPERATION

5.1 CYCLE START/FEED HOLD

● Cycle start (start automatic operation):

In the mode of automatic or inputting method, when automatic operation start signal ST is effective, programs start running.

1. Under the following conditions, ST signal is ignored:

A：any other methods except automatic method or inputting method
B：feed hold signal（SP）is 0
C：emergency stop signal（ESP）is 0
D：external reset signal（ERS）is 1
E：press “reset” button on the operator’s panel
F：CNC is warning
G：automatic operation has already started
H：program restart signal（SRN）is 1
I：CNC is searching for a sequence number

2. During automatic operation, CNC performs feed hold under the following conditions:

A：feed hold signal（SP）is 0
B：change to manual method, wheel method, single step method, machine zero return, or program zero return
C：single block command is finished
D：already finished inputting
E：CNC is warning
F：After changing to editing method, single block command is finished.

4、During automatic operation, CNC keeps in “reset” status and stops running under the following conditions:

A：emergency stop signal（ESP）is 0
B：external reset signal（ERS）is 1
C：press “reset” button on the operator’s panel

● Feed hold（pause automatic operation）:

During automatic operation, when the feed hold signal SP is 0, CNC pauses and stops running. At the same time cycle start indictor signal (STL) is set to 0, feed hold signal (SPL) is set to 1, even though SP is set to 1, the machine will not restart and operate automatically. Unless SP is set to 1, and ST signal is effective, the machine can restart and operate automatically.

When executing program blocks with only M, S, T commands, SP signal is set to 0, STL will be 0 immediately, SPL signal is 1, and CNC performs feed hold. When FIN signals are received from PLC, CNC continues executing the paused program blocks. After finish executing, SPL signal is 0 (STL signal is 0), CNC stops automatic operation.

While G92 command（thread cutting cycle）is executing, SP signal is 0, and SPL signal changes to 1 immediately, but the operation keeps running until cutter withdrawal is finished after thread cutting.

While G32 command (thread cutting) is executing, SP signal is 0, and SPL signal changes to 1 immediately, but the operation keeps running until the non-thread cutting program block is finished executing after the thread cutting block.

While G84 command (fixed tap cycle) is executing, SP signal is 0, SPL signal changes to 1 immediately, but the operation keeps running until the cutter returns to start position or R position after tap cutting is finished.

While executing macro command, after macro command is finished, it stops operating.

5.1.1 Cycle Start Signal

ST（G007.2）:

- Signal type: PLC→NC
- Signal function: under the automatic or editing method, PLC sets G7.2 to 1, and then sets to 0, and transmits to NC. NC starts automatic operation.
- Signal address:

G7						ST		

5.1.2 Feed Hold Signal

SP（G008.5）

- Signal type: PLC→NC
- Signal function: PLC sets G8.5 to 0, and transmits to NC. NC starts automatic operation. If G8.5 is 0, programs cannot restart.
- Signal address:

G8			SP					

5.1.3 Cycle Start Light Signal

STL（F000.5）

- Signal type: NC→PLC
- Signal function: when NC is on automatic operation, STL is set to 1 and transmitted to PLC.
- Signal address:

F0			STL					

5.1.4 Feed Hold Light Signal

SPL（F000.4）

- Signal type: NC→PLC
- Signal function: when NC is paused, SPL is set to 1 and transmitted to PLC.
- Signal address:

F0				SPL				

5.1.5 Automatic Operation Signal

OP（F000.7）

- Signal type: NC→PLC
- Signal function: when NC is on automatic operation, OP is set to 1 and transmitted to PLC. The relationship among STL, SPL and OP is as follows:

	Cycle start light STL	Feed hold light SPL	Automatic operation OP
Cycle start status	1	0	1
Feed hold status	0	1	1

Automatic operation status	0	0	0
Reset status	0	0	0

● Signal address:

F0	OP							

5.2 RESET

Under the following conditions, CNC starts "reset" status.

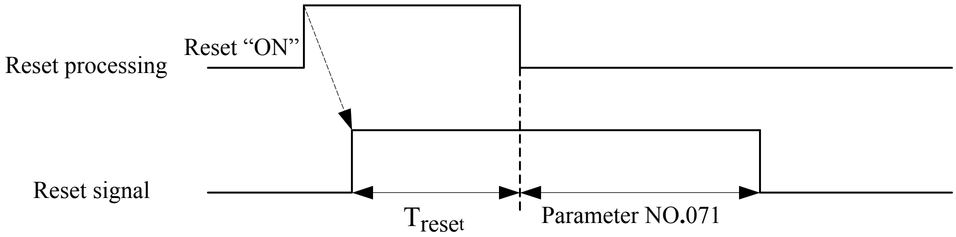
A：Emergency stop signal（ESP） is 0

B：External reset signal（ERS）is 1

C：Press the "reset" button on the panel

Except the above conditions, after the time specified by parameter NO.071, reset signal RST changes to 0.

RST time＝T_{reset}（reset processing time）＋reset signal output time（parameter NO.071）



During automatic operation, once CNC is reset, automatic operation stops, and the tool slows down along the controlled axis until stops in the end. If CNC is executing M, S, T commands, when it is reset, MF, SF, TF signals will change to 0 in 8ms.

5.2.1 External Reset Signal

ERS（G008.7）

● Signal type: PLC→NC

● Signal function: when PLC sets G8.7 to 1 and transmits to NC, CNC resets and RST signal becomes 1.

● Signal address:

G8	ERS							

5.2.2 Reset Signal

RST（F001.1）

● Signal type: NC→PLC

● Signal function: when NC is reset, it sets RST to 1 and transmits to PLC. This is to inform PLC that CNC is reset.

● Note: Under the following conditions, RST is set to 1:

A：external emergency stop signal (ESP) is set to 0.

B：external reset signal（ERS）is set to 1

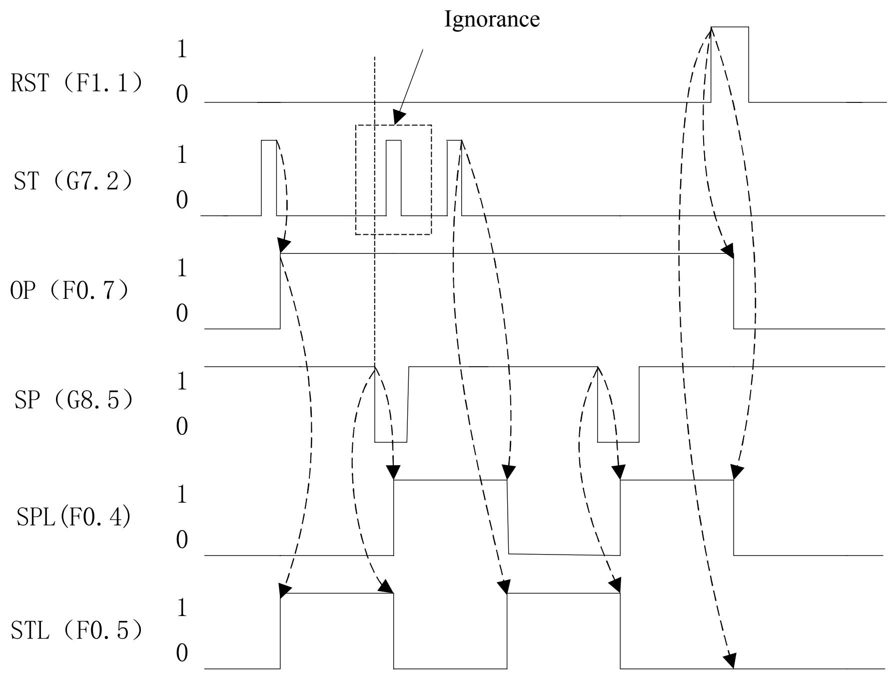
C：press the "reset" button on the panel

Except the above conditions, after the time specified by parameter NO.071, reset signal RST changes to 0.

● Signal address:

F1								RST	

5.2.3 Sequence of Start Operation



5.3 MACHINE LOCK

When checking programs before running the machine, machine lock function can be used. Set all-axis machine lock signal (MLK) to 1. In the process of manual operation or automatic operation, the machine stops pulse output to servo motor. But CNC still keeps command assignment, and absolute coordinates and relative coordinates are updated. Monitoring the changes of the coordinates positions can check if the edited programs are correct or not.

5.3.1 All-axis Machine Lock Signal

MLK（G44.1）

● Signal type: PLC →NC

● Signal functions: PLC sets G44.1 to 1, and transmits to NC. All axes are locked. Once G44.1 is set to 1, during the time of manual operation or automatic operation, CNC cannot output pulses to servo motor. Therefore the axis doesn't move.

● Signal address:

G44								MLK	

5.3.2 Detection Signal of All-axis Machine Lock

MMLK（F004.1）

● Signal type: NC→PLC

● Signal function: When all axes are locked, NC sets F4.1 to 1 and transmits to PLC.

● Signal address:

F4								MMLK

5.4 DRY DRUN

Dry run is effective in the automatic operation mode. At dry run time, the tool moves at an invariable override, instead of moving at a override specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed. The movement speed depends on manual feed override signal (JV0～JV15).

Rapid traverse selection signal （RT）	Dry run speed
1	Rapid traverse override
0	Manual feed override

5.4.1 Dry Run Signal

DRN（G46.7）：

● Signal type: PLC→NC

● Signal function: PLC sets G46.7 to 1, and transmits to NC. Then the dry run starts.

● Notes: A：If G46.7 is 1, the tool moves at a override specified for dry run. If G46.7 is 0, the tool moves normally.

B：When the tool is moving, and when G46.7 changes from 0 to 1 or from 1 to 0, the movement speed of the tool first drops down to 0, and then accelerates to a specified override.

● Signal address:

G46	DRN							

5.4.2 Detection Signal of Dry Run

MDRN（F002.7）：

● Signal type: NC→PLC

● Signal function: when a CNC is in the status of dry run, NC sets F2.7 to 1, and then transmits to PLC.

● Signal address:

F2	MDRN							

5.5 SINGLE BLOCK

Single block execution is effective in the automatic operation mode only. During the time of automatic operation, if single block signal（SBK）is set to 1, after the current block is executed, CNC starts feed hold. If single block signal（SBK） is set to 0, programs run normally.

5.5.1 Single Block Signal

SBK（G046.1）：

● Signal type: PLC→NC

● Signal function: When PLC sets G46.1 to 1, and transmits to NC. CNC starts single block executing.

● Signal address:

G46							SBK	

5.5.2 Detection Signal of Single Block

MSBK（F004.3）:

● Signal type: NC→PLC

● Signal function: when CNC is executing single block, NC sets F4.3 to 1 and transmits to PLC.

● Notes: A：thread cutting: during the time of thread cutting, SBK changes to 1. The executing stops until the first non thread cutting block is finished.

B：fixed cycle executing: during the time of fixed cycle executing, if SBK is set to 1, the executing stops when approaching drill and cutting withdrawal, instead of at the end of the program block.

● Signal address:

F4					**MSBK**			

5.6 OPTIONAL BLOCK JUMP

During automatic operation, if a slash "/" at the start of a block is specified, and the block jump signal BDT is set to 1, the program block is jumped and ignored.

5.6.1 Optional Block Jump Signal

BDT（G44.0）:

● Signal type: PLC→NC

● Signal function: If PLC sets G44.0 to 1 and transmits to NC, CNC starts executing block jumping. Any program block with a slash "/" at the start of the block will not executed.

● Signal address:

G44								**BDT**

5.6.2 Detection Signal of Jump Block

MBDT（F04.0）:

● Signal type: NC→PLC

● Signal function: when CNC is executing block jumping, NC sets F4.0 to 1 and transmits to PLC, which informs that PLC is running block jumping.

● Signal address:

F4								**MBDT**

Chapter 6　FEED SPEED CONTROL

6.1 RAPID TRAVERSE SIGNAL

The rapid traverse speed of each axis can be controlled by parameters NO.022 and NO.023, rather than program setting. But rapid traverse speed can also be controlled by adjusting rapid traverse override.

RPDO（F2.1）:

●Signal type：NC→PLC

●Signal function：When CNC is executing moving commands at a rapid traverse speed, NC sets F2.1 to 1, and transmits to PLC.

● Notes: A：If RPDO is 1, this indicates that an axis is moving after selecting rapid traverse mode; if RPDO is 0, it means an axis is moving after selecting null rapid traverse override.

B：In the automatic operation, rapid traverse includes all rapid traverse modes, such as fixed cycle position, machine zero return, etc, not only for G00 command. Manual rapid traverse also includes the rapid traverse in machine zero return.

C：Once rapid traverse is selected, the signal keeps to 1, even during the stop time, but changes until other feed speed is selected and moving is started.

●Signal address

F2								
							RPDO	

6.2 RAPID TRAVERSE OVERIRDE

Rapid traverse override has four levels F0, 25%, 50% and 100%. F0 override is specified by parameter NO.032.

In automatic method or manual method (including machine zero return, program zero return), actual moving rate is obtained through multiplying the value set by parameters NO.022, NO.023 by the rate value

6.2.1 Rapid Traverse Override Signal

ROV1、ROV2（G014.0、G014.1）

●Signal type：PLC→NC

●Signal function：PLC sets values to G14.0, G14.1 and transmits to NC. NC then sets the rapid traverse override according to the values of G14.0, G14.1. The override values of ROV1, ROV2 are shown in the following table:

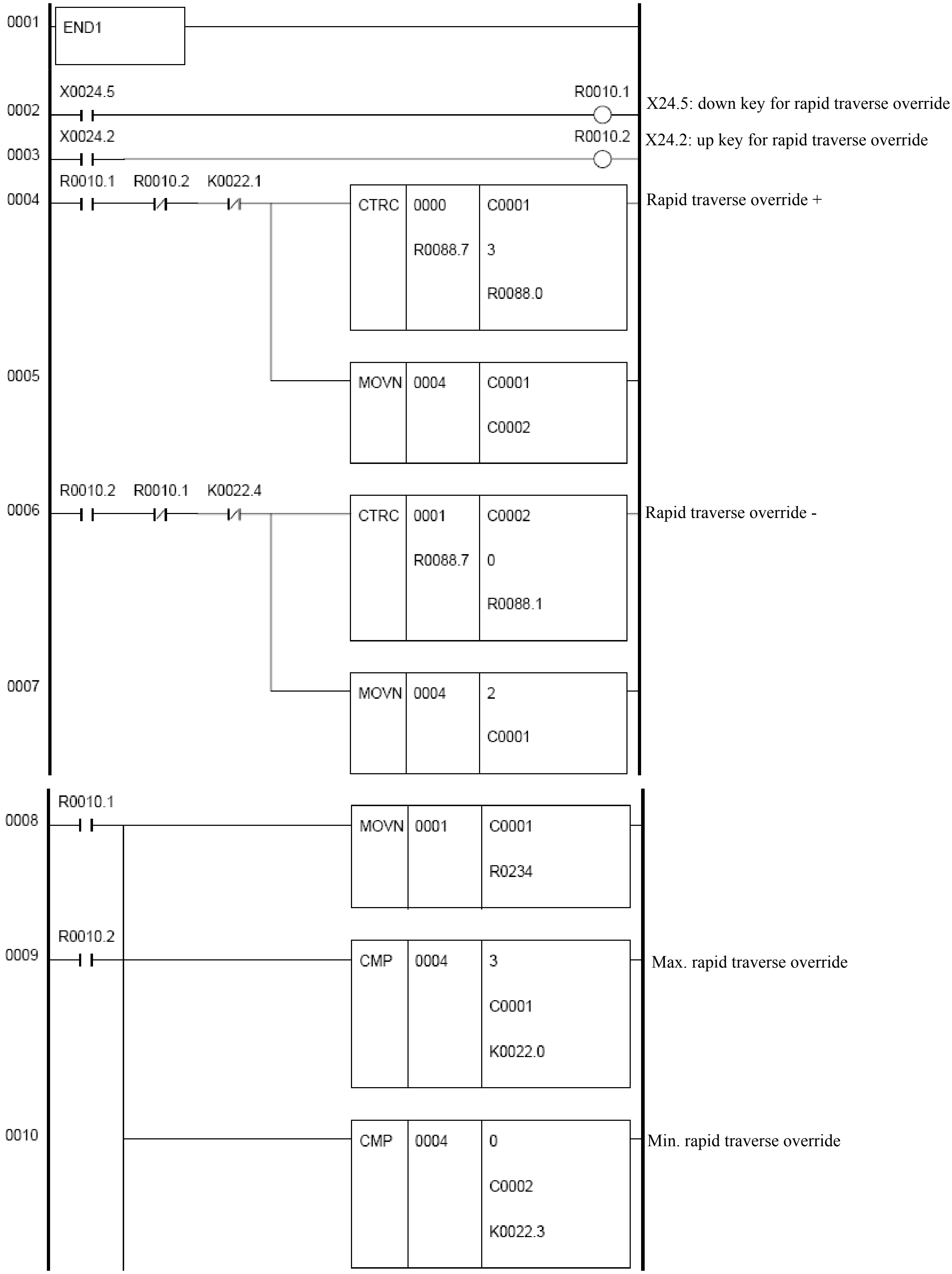
ROV2（G14.1）	ROV1（G14.0）	Override value
0	0	100％
1	0	50％
0	1	25％
1	1	FO

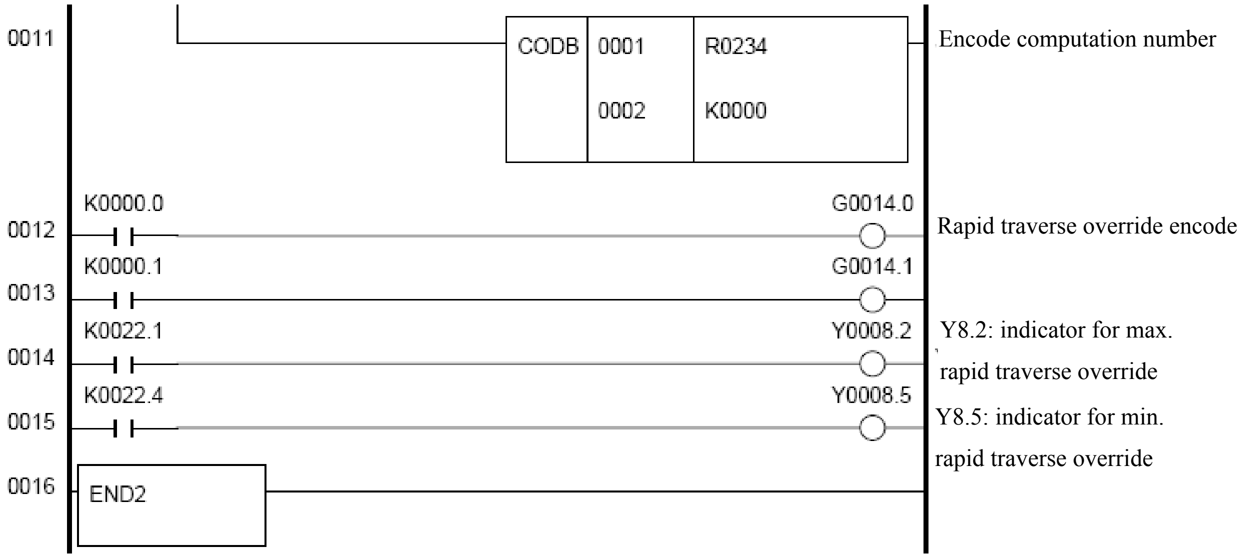
●Signal address

G14								
							ROV2	ROV1

Book 2 Function

6.2.2 Rapid Traverse Override Control Flowchart





6.3 FEED OVERRIDE

Feed speed can be adjusted by feed override, and can be used for checking programs. For example, if the feed speed specified in the program is 100mm/min, when the feed override is adjusted to 50％, the tool moves at a speed of 50mm/min.

6.3.1 Feed Override Signal

FV0～FV7 (G012):

●Signal type：PLC→NC

●Signal function：PLC sets a value to G12 and transmits to NC. NC can set a feed override according to the value of G12. The relationship between FV0 - FV7 binary codes and feed override is shown below:

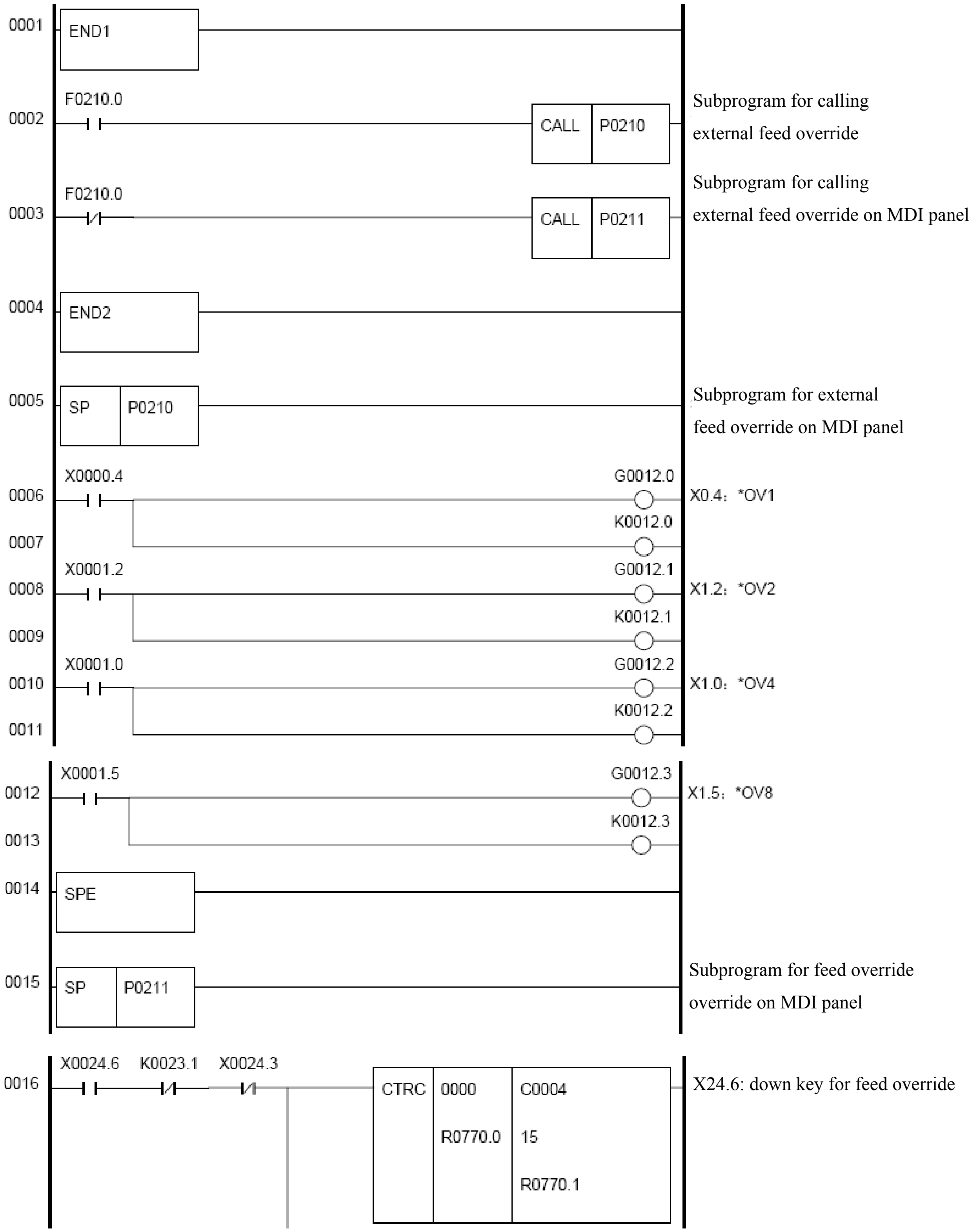
FV7～FV0　（G012.7～G012.0)	Cutting feed override
0000 1111	0％
0000 1110	10％
0000 1101	20％
0000 1100	30％
0000 1011	40％
0000 1010	50％
0000 1001	60％
0000 1000	70％
0000 0111	80％
0000 0110	90％
0000 0101	100％
0000 0100	110％
0000 0011	120％
0000 0010	130v
0000 0001	140％
0000 0000	150％

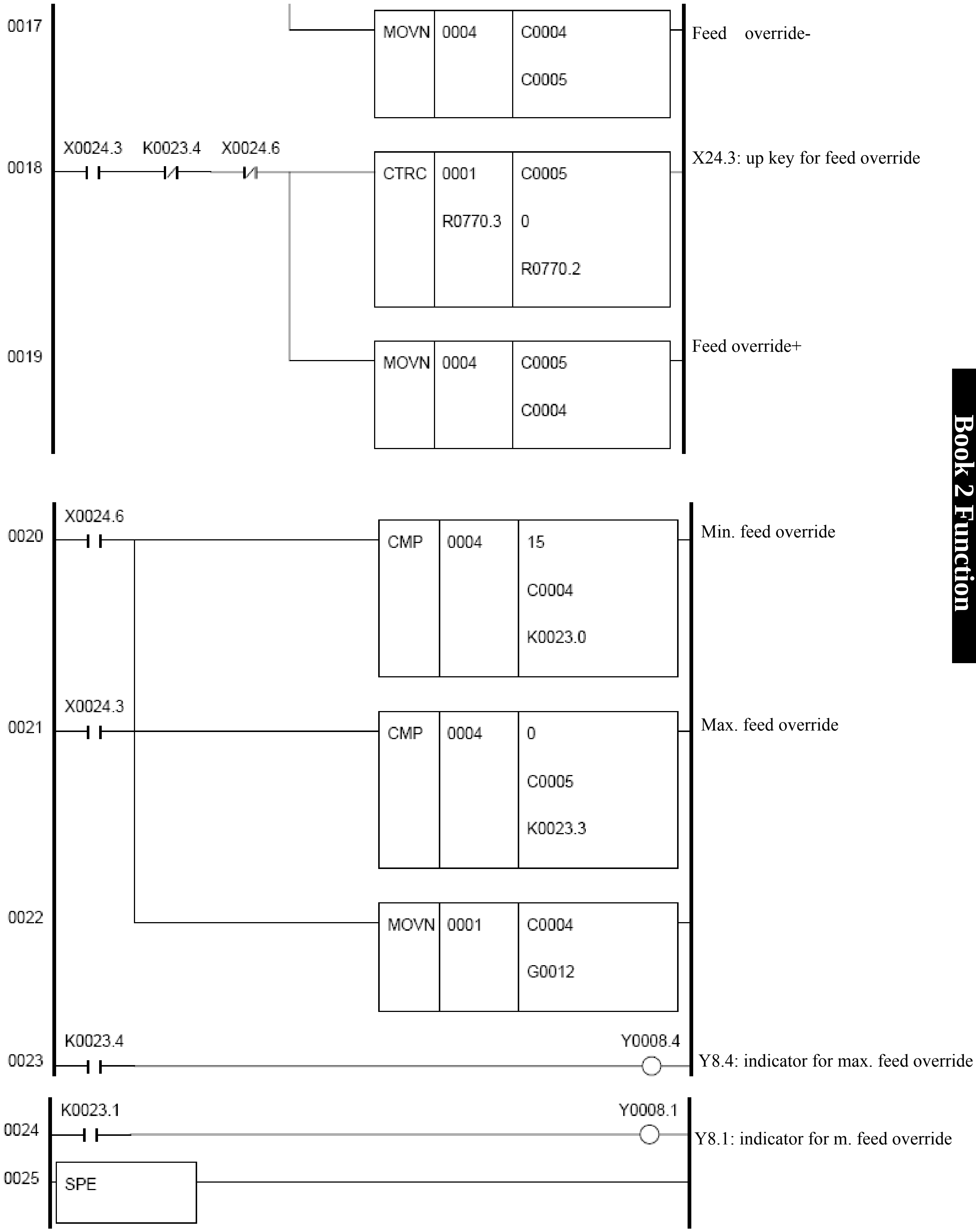
● Note: In automatic operation, actual feed speed = cutting feed speed x feed override.

●Signal address

G12	FV7	FV6	FV5	FV4	FV3	FV2	FV1	FV0

6.3.2 Feed Override Control Flowchart





Book 2 Function

6.4 OVERRIDE CANCEL SIGNAL

OVC（G006.4）:

●Signal type：PLC→NC

●Signal function：PLC sets G6.4 to 1 and transmits to NC. NC will set the feed override to be 100%.

●Note：when OVC is 1, CNC operates as follows:

A：no matter how the feed override signal is adjusted, the feed override remains unchanged 100％

B：rapid traverse override and spindle override are not affected.

●Signal address：

G6				**OVC**				

Chapter 6   FEED SPEED CONTROL

6.1 RAPID TRAVERSE SIGNAL

The rapid traverse speed of each axis can be controlled by parameters NO.022 and NO.023, rather than program setting. But rapid traverse speed can also be controlled by adjusting rapid traverse override.

RPDO（F2.1）:

●Signal type：NC→PLC

●Signal function：When CNC is executing moving commands at a rapid traverse speed, NC sets F2.1 to 1, and transmits to PLC.

● Notes: A：If RPDO is 1, this indicates that an axis is moving after selecting rapid traverse mode; if RPDO is 0, it means an axis is moving after selecting null rapid traverse override.

B：In the automatic operation, rapid traverse includes all rapid traverse modes, such as fixed cycle position, machine zero return, etc, not only for G00 command. Manual rapid traverse also includes the rapid traverse in machine zero return.

C：Once rapid traverse is selected, the signal keeps to 1, even during the stop time, but changes until other feed speed is selected and moving is started.

●Signal address

F2								
							RPDO	

6.2 RAPID TRAVERSE OVERIRDE

Rapid traverse override has four levels F0, 25%, 50% and 100%. F0 override is specified by parameter NO.032.

In automatic method or manual method (including machine zero return, program zero return), actual moving rate is obtained through multiplying the value set by parameters NO.022, NO.023 by the rate value

6.2.1 Rapid Traverse Override Signal

ROV1、ROV2（G014.0、G014.1）

●Signal type：PLC→NC

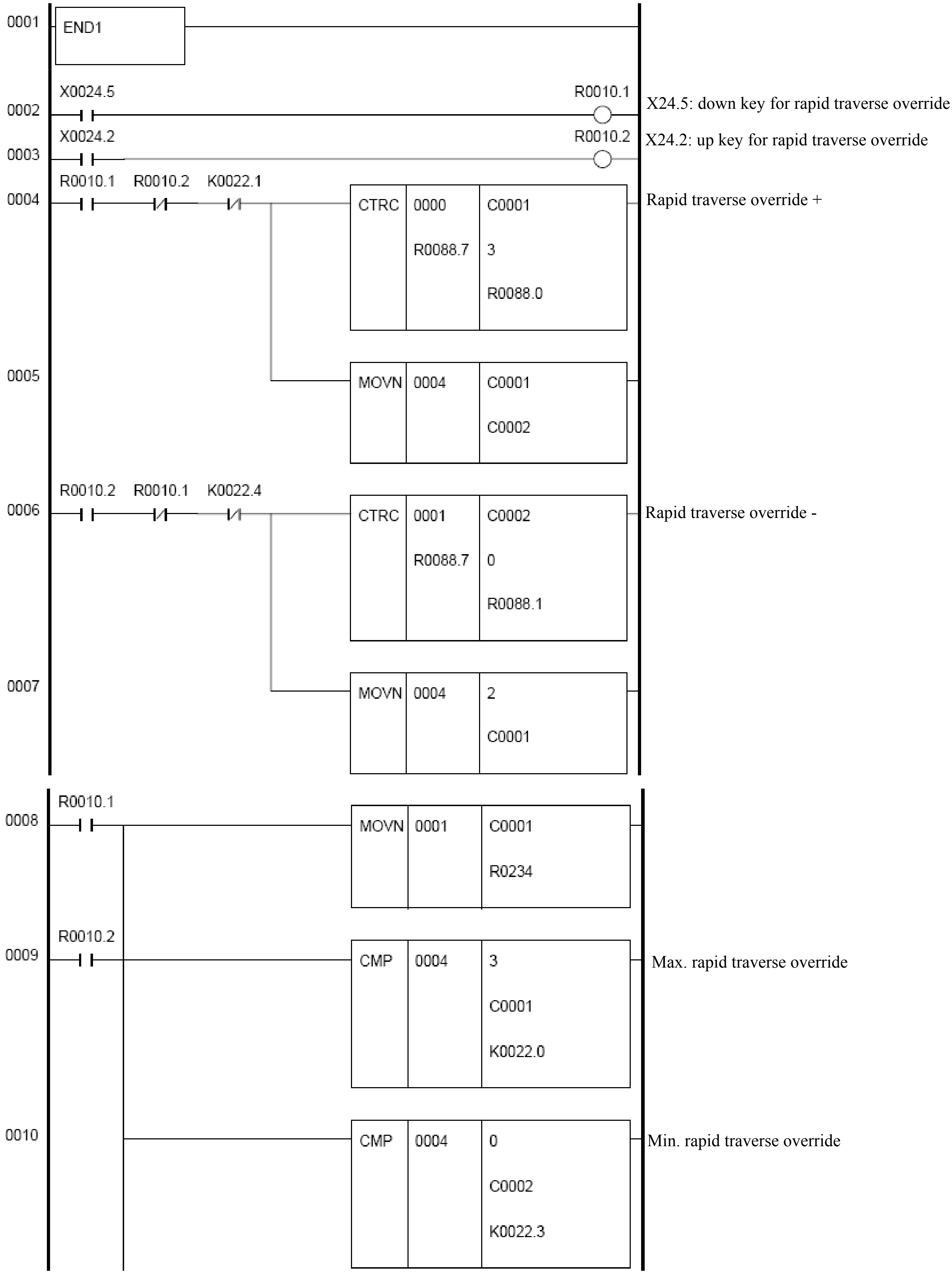
●Signal function：PLC sets values to G14.0, G14.1 and transmits to NC. NC then sets the rapid traverse override according to the values of G14.0, G14.1. The override values of ROV1, ROV2 are shown in the following table:

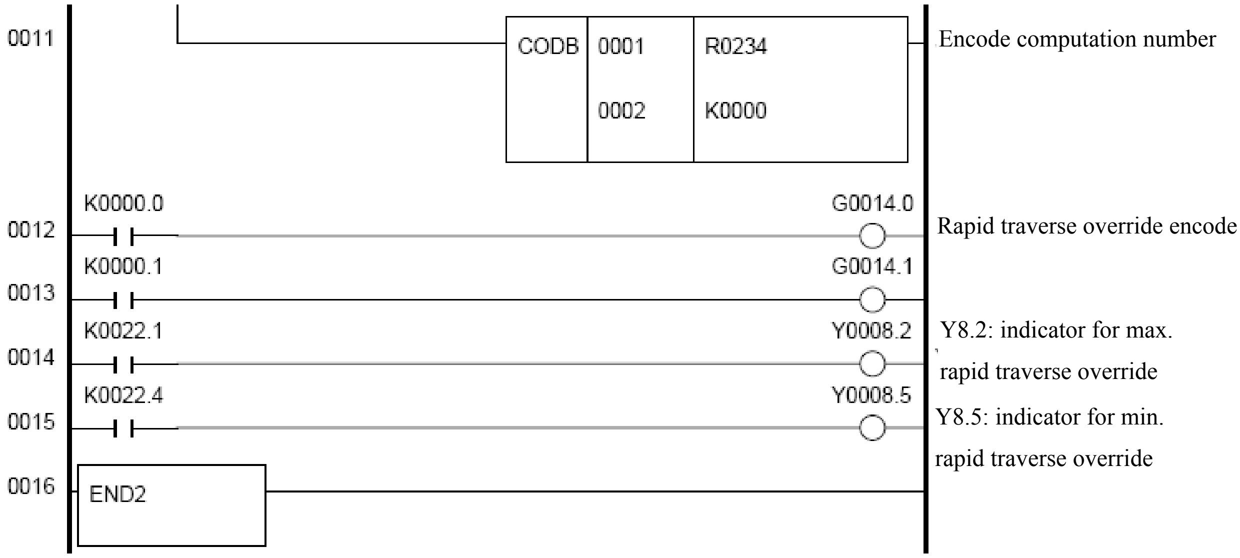
ROV2（G14.1）	ROV1（G14.0）	Override value
0	0	100％
1	0	50％
0	1	25％
1	1	FO

●Signal address

G14								
							ROV2	**ROV1**

6.2.2 Rapid Traverse Override Control Flowchart





6.3 FEED OVERRIDE

Feed speed can be adjusted by feed override, and can be used for checking programs. For example, if the feed speed specified in the program is 100mm/min, when the feed override is adjusted to 50％, the tool moves at a speed of 50mm/min.

6.3.1 Feed Override Signal

FV0～FV7 (G012):

●Signal type：PLC→NC

●Signal function：PLC sets a value to G12 and transmits to NC. NC can set a feed override according to the value of G12. The relationship between FV0 - FV7 binary codes and feed override is shown below:

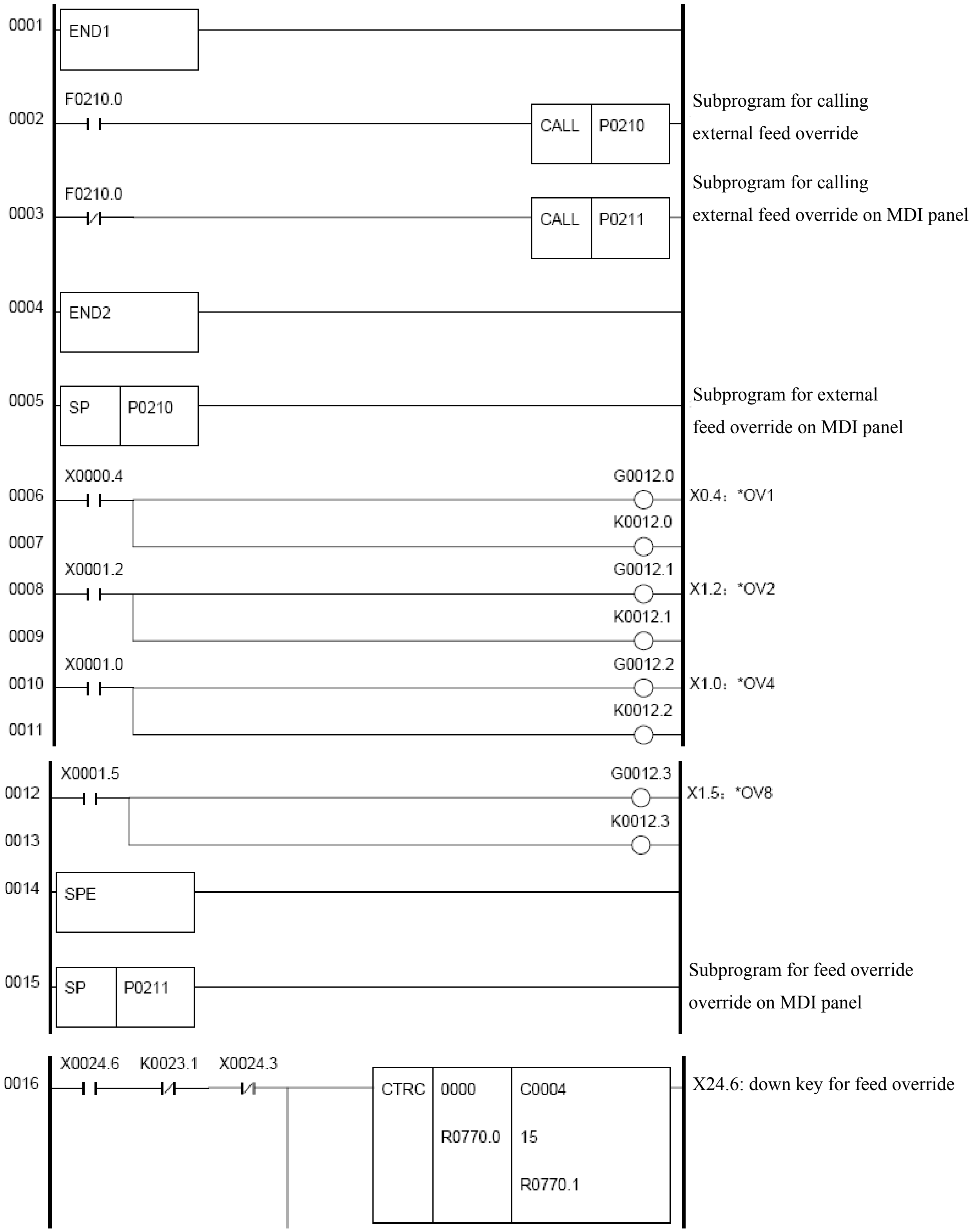
FV7～FV0 （G012.7～G012.0)	Cutting feed override
0000 1111	0％
0000 1110	10％
0000 1101	20％
0000 1100	30％
0000 1011	40％
0000 1010	50％
0000 1001	60％
0000 1000	70％
0000 0111	80％
0000 0110	90％
0000 0101	100％
0000 0100	110％
0000 0011	120％
0000 0010	130v
0000 0001	140％
0000 0000	150％

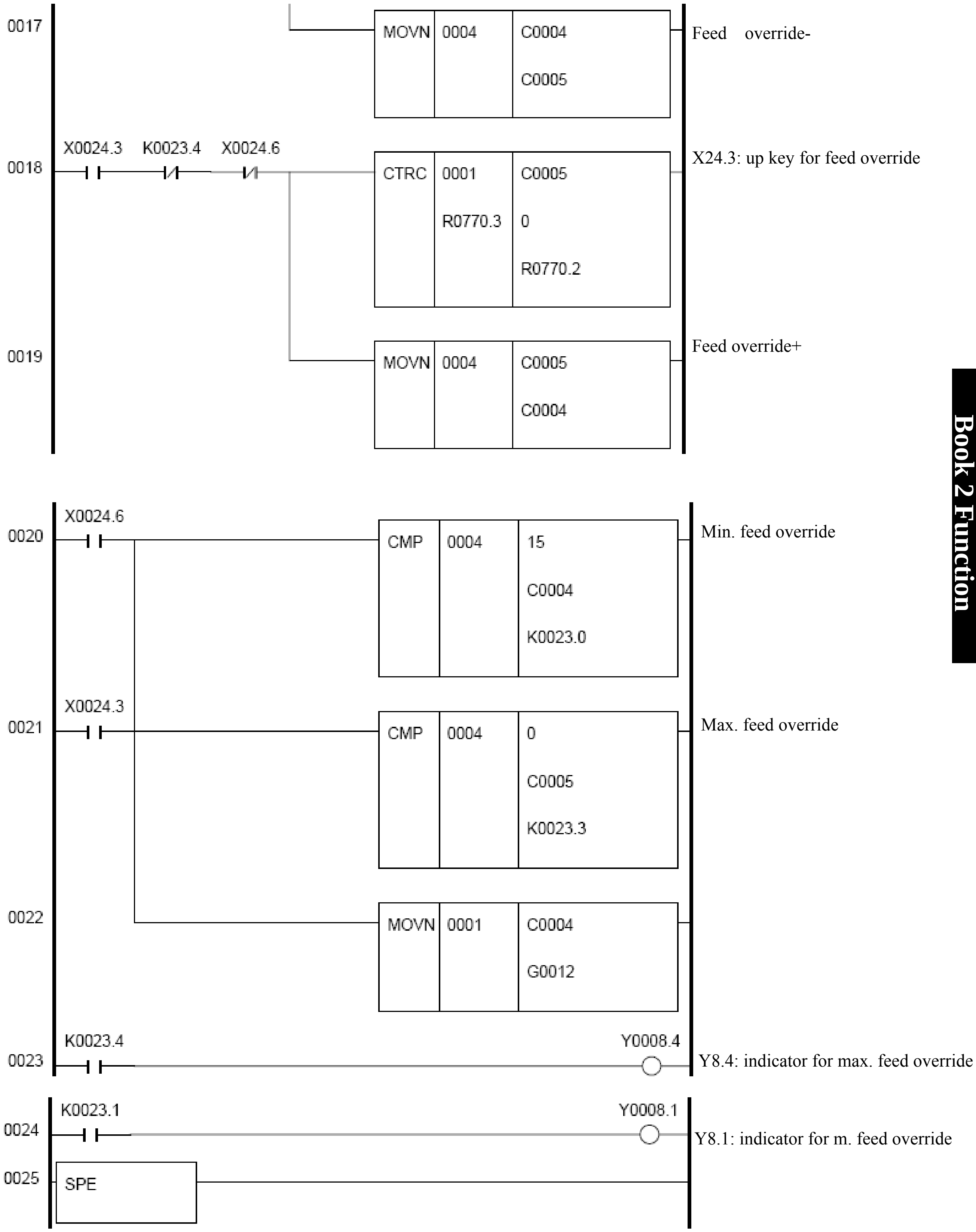
● Note: In automatic operation, actual feed speed = cutting feed speed x feed override.

●Signal address

G12		FV7	FV6	FV5	FV4	FV3	FV2	FV1	FV0

6.3.2 Feed Override Control Flowchart





6.4 OVERRIDE CANCEL SIGNAL

OVC（G006.4）：

●Signal type：PLC→NC

●Signal function：PLC sets G6.4 to 1 and transmits to NC. NC will set the feed override to be 100%.

●Note：when OVC is 1, CNC operates as follows:

A：no matter how the feed override signal is adjusted, the feed override remains unchanged 100％

B：rapid traverse override and spindle override are not affected.

●Signal address：

G6								
				OVC				

Chapter 7　M S T Functions

When a maximum 8-digit number following M, S, T address is specified, the corresponding code signals and strobe signals are sent to PLC. PLC starts logic control according to these signals, as shown below:

Function	Program address	NC→PLC			End signal (PLC→NC)
		Code signal	Strobe signal	Distribution of end signal	
Miscellaneous function	M	M00～M31（F10～F13）	MF（F7.0）	DEN（F1.3）	FIN（G4.3）
Spindle speed function	S	S00～S31（F22～F25）	SF（F7.2）		
Tool function	T	T00～T31（F26～F29）	TF（F7.3）		

The process is as follows: (change M code to S, T code, i.e. the process of spindle speed function and tool function)

A：Supposed M_ _ is specified in the program, while CNC doesn't specify, an alarm occurs.

B：After sending code signals M00 - M31 to PLC, and strobe signal MF is set to 1, code signal uses binary system to express program command value _ _. If moving is paused, spindle speed or other functions are commanded together with auxiliary functions, after code signals of the auxiliary functions are sent, other functions start executing.

C：When strobe signal MF is 1, PLC reads the code signal and executes corresponding operations.

D：If move command and pause command is included in a program block, in order to execute auxiliary functions after move commands and pause commands are finished, have to wait until DEN signal changes to 1.

E：When operation stops, PLC sets the end signal FIN to 1. End signal is used for auxiliary functions, spindle speed functions and tool functions. If all these functions are executed at the same time, end signal FIN changes to 1 until all functions are stopped.

F：When end signal FIN becomes 1 for a period of time, CNC can set the strobe signal to 0 and confirm end signal is received.

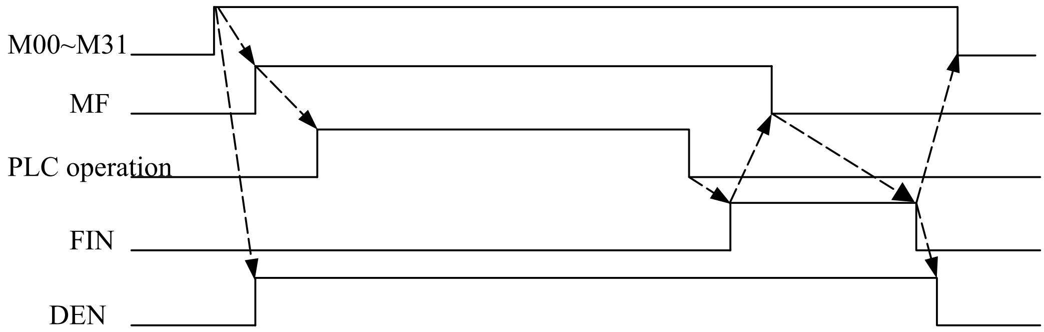
G：Until strobe signal is 0, PLC can set FIN to 0.

H：When FIN is 0, CNC sets all code signals to 0 and stops all the sequence operation of the auxiliary function. (While executing spindle speed function and tool function, code signal keeps unchanged until a new code is specified.)

I：After commands in a same block is finished, CNC starts executing next block.

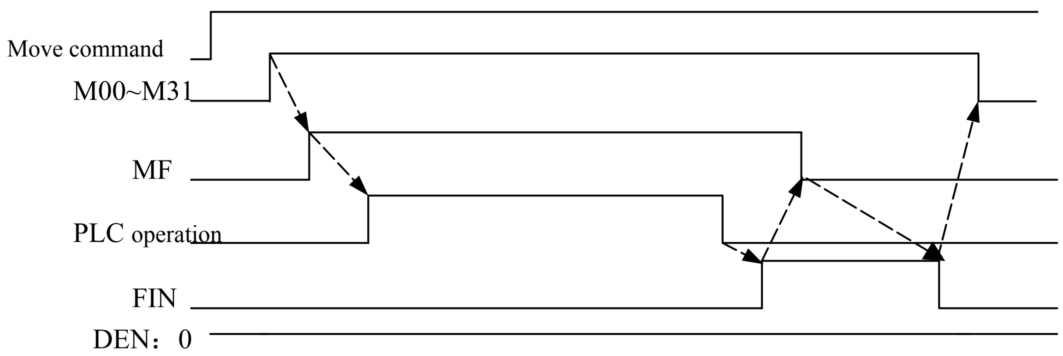
In application, the following sequences can be selected as per logical requirements:

If there is one auxiliary function in the block, PLC uses the following sequence:

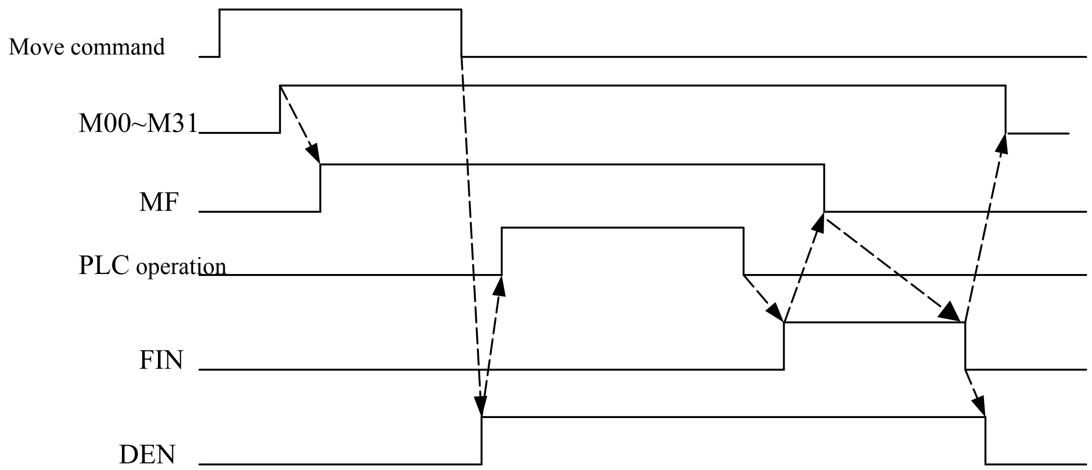


When move command and auxiliary function are in a same block, if PLC executes auxiliary function before move command ends, it executes in the following sequence:

Book 2 Function



When move command and auxiliary function are in a same block, if PLC executes auxiliary function after move command ends, it executes in the following sequence:



7.1 AUXILIARY FUNCTION（M FUNCTION）

7.1.1 Code Signal and Strobe Signal

Code signal: M00～M31（F010～F013）, strobe signal: MF（F007.0）

●Signal type: NC→PLC

●Signal functions: After CNC executes M code, NC sends M code to PLC via F10 - F13, and sets MF to 1, to start logical control. For related output conditions and executing processes, please refer to above descriptions. The relationship between M command and code signal number is as follows:

F013，F012，F011，F010	M command
00000000，00000000，00000000，00000000	M00
00000000，00000000，00000000，00000001	M01
00000000，00000000，00000000，00000010	M02
00000000，00000000，00000000，00000011	M03
00000000，00000000，00000000，00000100	M04
00000000，00000000，00000000，00000101	M05
00000000，00000000，00000000，00000110	M06
00000000，00000000，00000000，00000111	M07
00000000，00000000，00000000，00001000	M08
.	.

● Notes: The following auxiliary function commands cannot be output even if they are instructed in CNC programs:

A：M98，M99

B：M code for subprogram call

C：M code for custom macro program call

●Signal addresses:

F10	**M07**	**M06**	**M05**	**M04**	**M03**	**M02**	**M01**	**M00**
F11	**M15**	**M14**	**M13**	**M12**	**M11**	**M10**	**M09**	**M08**
F12	**M23**	**M22**	**M21**	**M20**	**M19**	**M18**	**M17**	**M16**
F13	**M31**	**M30**	**M29**	**M28**	**M27**	**M26**	**M25**	**M24**
F7								**MF**

7.1.2 M Code Signal

DM00（F009.7）、DM01（F009.6）、DM02（F009.5）、DM30（F009.4）:

●Signal type：NC→PLC

●Signal function：When CNC is executing M00, M01, M02, M30 commands, NC sets corresponding code signals DM00, DM01, DM02, DM30 to 1.

Program command	Code signal
M00	DM00
M01	DM01
M02	DM02
M30	DM30

● Notes:　1：Under the following conditions, M code signal is 1: related auxiliary function is specified, and other move commands and pause commands are finished in a same block. (If NC receives FIN signal before move command and pause command end, M code signal will not be outputted.)

2：When FIN signal is 1 or the tool is reset, M code signal is 0.

3：While executing M00, M01, M02, M30, code signals DM00, DM01, DM02, DM30 and code signals M00 - M31 and strobe signal MF are all outputted at the same time.

●Signal address

F9	**DM00**	**DM01**	**DM02**	**DM30**				

7.2 Spindle Speed Function (S Function)

Spindle speed code signal S00 - S31（F022 - F025）, spindle speed strobe signal SF（F007.2）

●Signal type：NC→PLC

●Signal function：After CNC executes S code, NC sends S code to PLC via F22 – F25, and then sets SF to 1, and transmits to PLC for logical control. For output conditions and executing progresses, please refer to related description. The relationship between S command and binary code signal is shown in the below table:

F025，F024，F023，F022	S command
00000000，00000000，00000000，00000000	S00
00000000，00000000，00000000，00000001	S01
00000000，00000000，00000000，00000010	S02
00000000，00000000，00000000，00000011	S03
00000000，00000000，00000000，00000100	S04
. . .	. . .

●Signal addresses:

F22	**S07**	**S06**	**S05**	**S04**	**S03**	**S02**	**S01**	**S00**
F23	**S15**	**S14**	**S13**	**S12**	**S11**	**S10**	**S09**	**S08**
F24	**S23**	**S22**	**S21**	**S20**	**S19**	**S18**	**S17**	**S16**
F25	**S31**	**S30**	**S29**	**S28**	**S27**	**S26**	**S25**	**S24**
F7						**SF**		

7.3 Tool Function (T Function）

Tool function code signal T00 - T31（F026 - F029）, tool function strobe signal TF（F007.3）

●Signal type：NC→PLC

●Signal function：After CNC executes T code, NC sends T code to PLC via F26 - F29, and then sets TF to 1 and transmits to PLC for logical control. For output conditions and executing progresses, please refer to related description. The relationship between T command and binary code signal is shown in the below table:

F029，F028，F027，F026	T command
00000000，00000000，00000000，00000000	T00
00000000，00000000，00000000，00000001	T01
00000000，00000000，00000000，00000010	T02
00000000，00000000，00000000，00000011	T03
00000000，00000000，00000000，00000100	T04
. . . .	. . . .

●Signal addresses:

F26	**T07**	**T06**	**T05**	**T04**	**T03**	**T02**	**T01**	**T00**
F27	**T15**	**T14**	**T13**	**T12**	**T11**	**T10**	**T09**	**T08**
F28	**T23**	**T22**	**T21**	**T20**	**T19**	**T18**	**T17**	**T16**
F29	**T31**	**T30**	**T29**	**T28**	**T27**	**T26**	**T25**	**T24**
F7					**TF**			

7.4 MST Function End

7.4.1 End Signal

FIN（G004.3）

●Signal type：PLC→NC

●Signal function：When auxiliary functions, spindle speed function and tool function are all finished executing, PLC sets FIN to 1 and transmits to NC.

●Signal address：

G4				**FIN**				

7.4.2 Distribution End Signal

DEN（F001.3）：

●Signal type：NC→PLC

●Signal function：If auxiliary functions, spindle speed function, tool function and other commands (e.g. move command, pause command) are all in a single block, after other commands are finished executing, waiting for FIN signal, then NC sets DEN to 1. Every time a block ends executing, DEN will be set to 0.

●Signal address:

F1				**DEN**				

7.5 AUNCTION LOCK SIGNAL

7.5.1 Auxiliary Function Lock Signal

AFL（G0005.6）：

●Signal type：PLC→NC

●Signal function：PLC sets G5.6 to 1, then transmits to NC to disenable the execution of M, S, T functions.

● Notes: When AFL signal is 1, CNC works as below:

Book 2 Function

A：For operations with automatic method and inputting method, CNC doesn’t execute specified M, S and T functions, i.e. code signal and strobe signal are not outputted.

B：If AFL is set to 1 after code signal is outputted, CNC works in a normal way until execution finishes (until FIN signal is received and strobe signal is set to 0).

C：When AFL is 1, M00, M01, M02 and M30 commands can be executed, and corresponding code signal, strobe signal, coding signal are all outputted in a normal way.

D：If AFL is 1, auxiliary functions M98 and M99 still works in a normal way, but no results are outputted.

E：If AFL is 1, spindle analog value can be outputted.

●Signal address：

G5		AFL						

7.5.2 Detection Signal of Auxiliary Function Lock

MAFL（F004.4）:

●Signal type：NC→PLC

●Signal function：When CNC is executing auxiliary function lock, NC sets F4.4 to 1 and transmits to PLC.

●Signal address:

F4				MAFL				

Chapter 8　SPINDLE SPEED

The spindle is divided into gear spindle and analog spindle according its control mode. CNC converts S code into switching value to output to spindle to control spindle speed in gear spindle mode; CNC converts S code into analog value to spindle to control spindle speed in analog spindle mode; CNC transmits SIMSPL signal to inform PLC of current spindle control mode.

Spindle mode signal SIMSPL（F200.4）

●Signal type：NC→PLC

●Signal function: NC sets SIMSPL to 0 or 1, sets control mode of spindle speed，transmits SIMSPL to PLC and informs PLC of current spindle speed control mode.

●Signal action: SIMSPL=1: CNC sets analog spindle control; S code is converted into analog value and is transmitted.

SIMSPL=0: CNC sets gear spindle control; S code is converted into switching value and is transmitted.

●Signal address

F200				SIMSPL				

8.1 GEAR SPINDLE

Gear spindle is defined that the actual spindle speed is controlled by gears of machine, and CNC converts S code into switching value to control gears of machine to realize spindle speed control. Refer to ***Section 7.2.***

8.2 ANALOG SPINDLE

8.2.1 Analog Spindle Speed Control

S instruction for analog spindle control: it is input by part programs and used for specifying analog spindle speed controlled by CNC. For constant surface speed cutting (G96), CNC converts the specified constant surface speed into spindle speed.

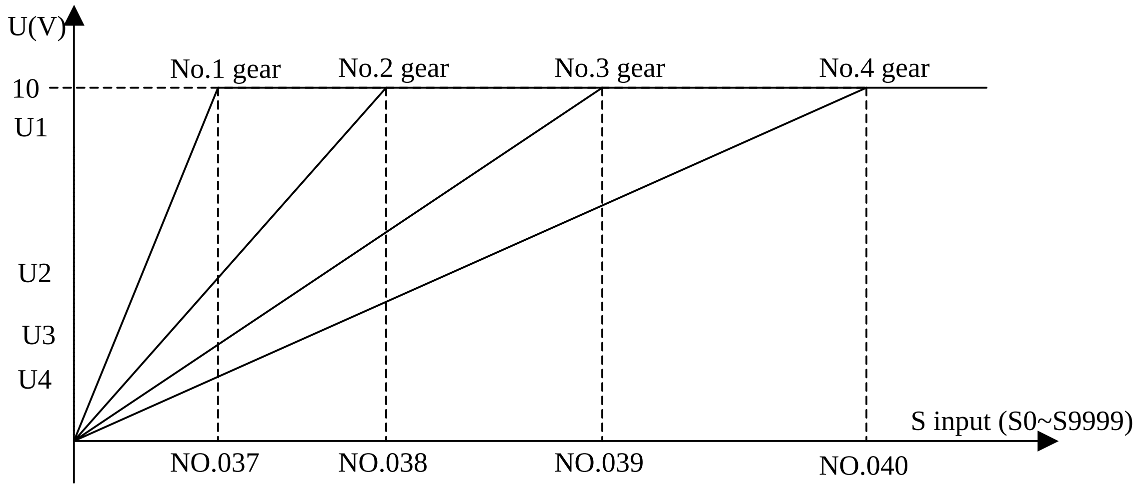
S code/SF signal output: analog spindle control function in CNC outputs S instruction with binary code to PLC and does not output SF.

Gear shifting: although S instruction specifies spindle speed, actual control of CNC is spindle motor. CNC outputs the corresponding spindle speed when the machine gear shifts to some gear.

Gear shifting method: to execute gear shifting, max. spindle speed corresponding to each gear is set by NO.037～NO.040. Selection signal of gear shift is 2-bit code signal(GR1, GR2). Relationship between signal and gear is as follows:

GR2(G28.2)	GR1(G28.1)	Gear	Parameter number of max. spindle speed
0	0	1	Data parameter NO.037
0	1	2	Data parameter NO.038
1	0	3	Data parameter NO.039
1	1	4	Data parameter NO.040

NO.037～NO.040 separately set No. 1~No. 4 max. spindle speed. When analog voltage 10V corresponds to max. speed of spindle motor，linear relationship of analog voltage output for the same S instruction in different gears is as follows：



Suppose that NO.037＝1000；NO.038＝2000；NO.039=4000；NO.040=5000. S=800:

G28.1=0, G28.2=0, No. 1 gear:

$$U1 = \frac{800}{1000} \times 10 = 8V$$

G28.1=0, G28.2=0, No. 2 gear:

$$U2 = \frac{800}{2000} \times 10 = 4V$$

G28.1=0, G28.2=0, No. 3 gear:

$$U3 = \frac{800}{4000} \times 10 = 2V$$

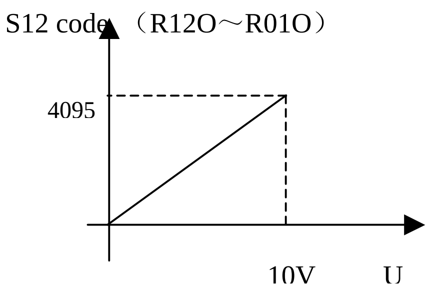
G28.1=0, G28.2=0, No. 4 gear:

$$U4 = \frac{800}{5000} \times 10 = 1.6V$$

Formula operation of output voltage value:

$$U = \frac{\text{S instruction}}{\text{Max.speed of current gear}} \text{X}10$$

S12 code (R12O～R01O) output: NC counts the spindle speed of each gear through gear shifting control, namely S12 code signal R12O～R01O（0~4095）output the spindle motor and PLC. Relationship between 10V and S12 code is as follows:

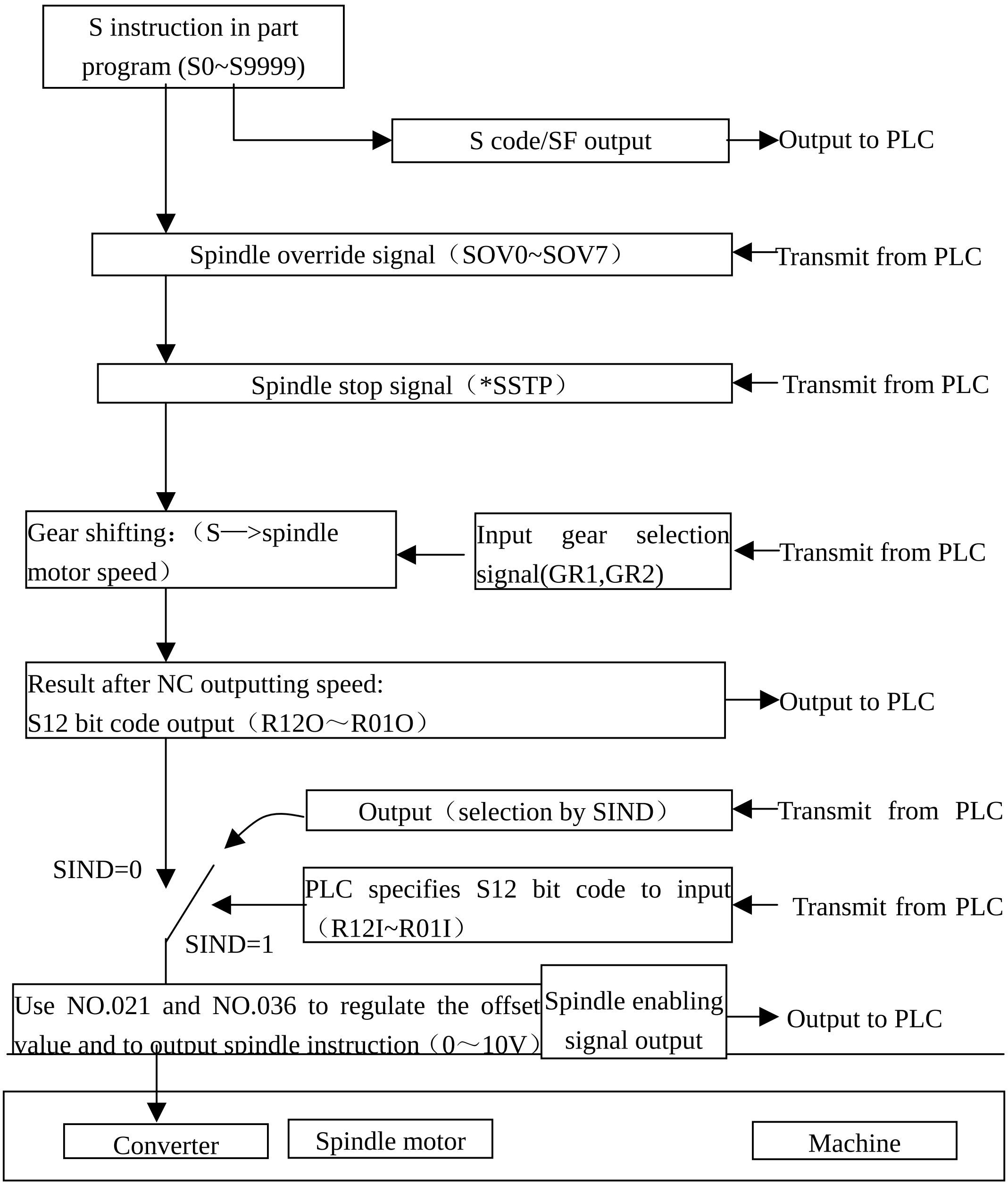


S12 code output: SIND (G33.7) determinates the spindle speed instruction uses S12 code data（R12Ｏ～R01Ｏ）counted by NC or S12 bit code signal (spindle motor speed instruction signal R12I～R01I) specified by PLC. The spindle speed instruction is set in R12I～R01I if the spindle speed is transmitted from PLC.

Voltage offset: after the system receives the spindle speed S12 bit code（0~4095）, it compensates and regulates instruction value of spindle speed by setting NO.021 (it is voltage compensation value when the analog voltage outputs 0V) and NO.036 (it is voltage compensation value when the analog voltage outputs 0V). Operation formula is as follows:

$$\text{Output value of spindle speed} = \frac{9999 + NO.021 - NO.036}{9999} \times S12\ \text{code} + NO.036 \times \frac{4095}{9999}$$

Conditions of stop output: when S0 is commanded (*SSTP＝0 or S0), the instruction output to spindle is reset to 0. An instruction from NC to spindle cannot be reset to 0 with M05, emergent stop or reset mode.
Analog spindle interface: the system outputs analog voltage of the regulated spindle speed to the spindle motor by analog spindle interface circuit.
Control instruction flow of analog spindle speed



Spindle stop signal *SSTP（G29.6）:

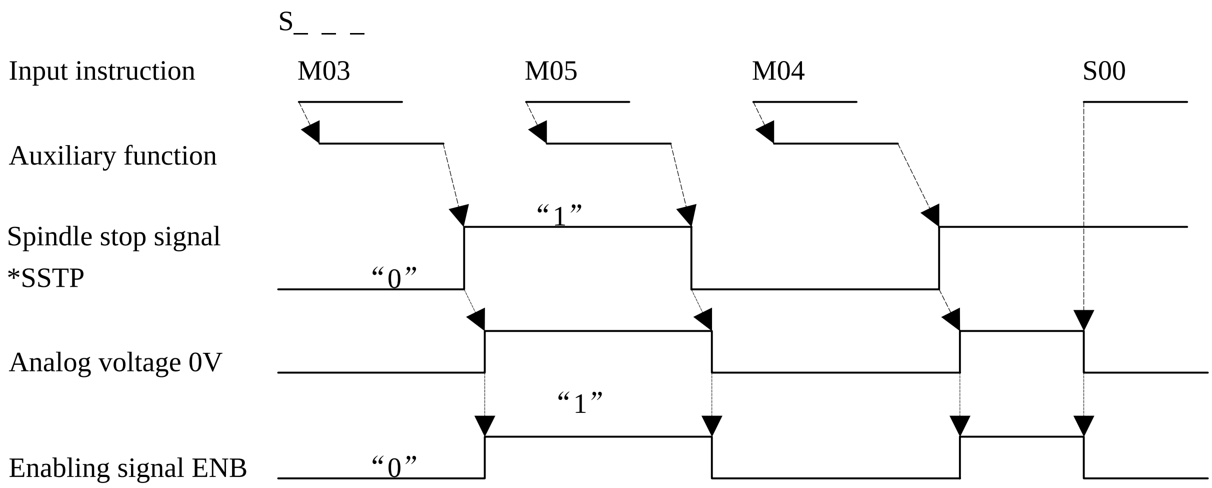
●Signal type：PLC→NC

●Signal function: stop output the spindle speed instruction. The signal sets S instruction in NC to be 0. Its relative time sequence is as follows:

●Caution: when the spindle stop signal ***SSTP** is 0, the output voltage is 0V. When the signal is 1, the analog voltage outputs instruction value. When the signal is not used, the signal is set to 1 to make CNC control spindle speed.

●Signal address：

G29								
		***SSTP**						



Spindle speed override signal SOV0～SOV7（G30）

●Signal type：PLC→NC

●Signal function: PLC evaluates G30 to transmit it to NC which will determine the spindle speed override according to G30 value. Relation between SOV0～SOV7 and override value is as follows:

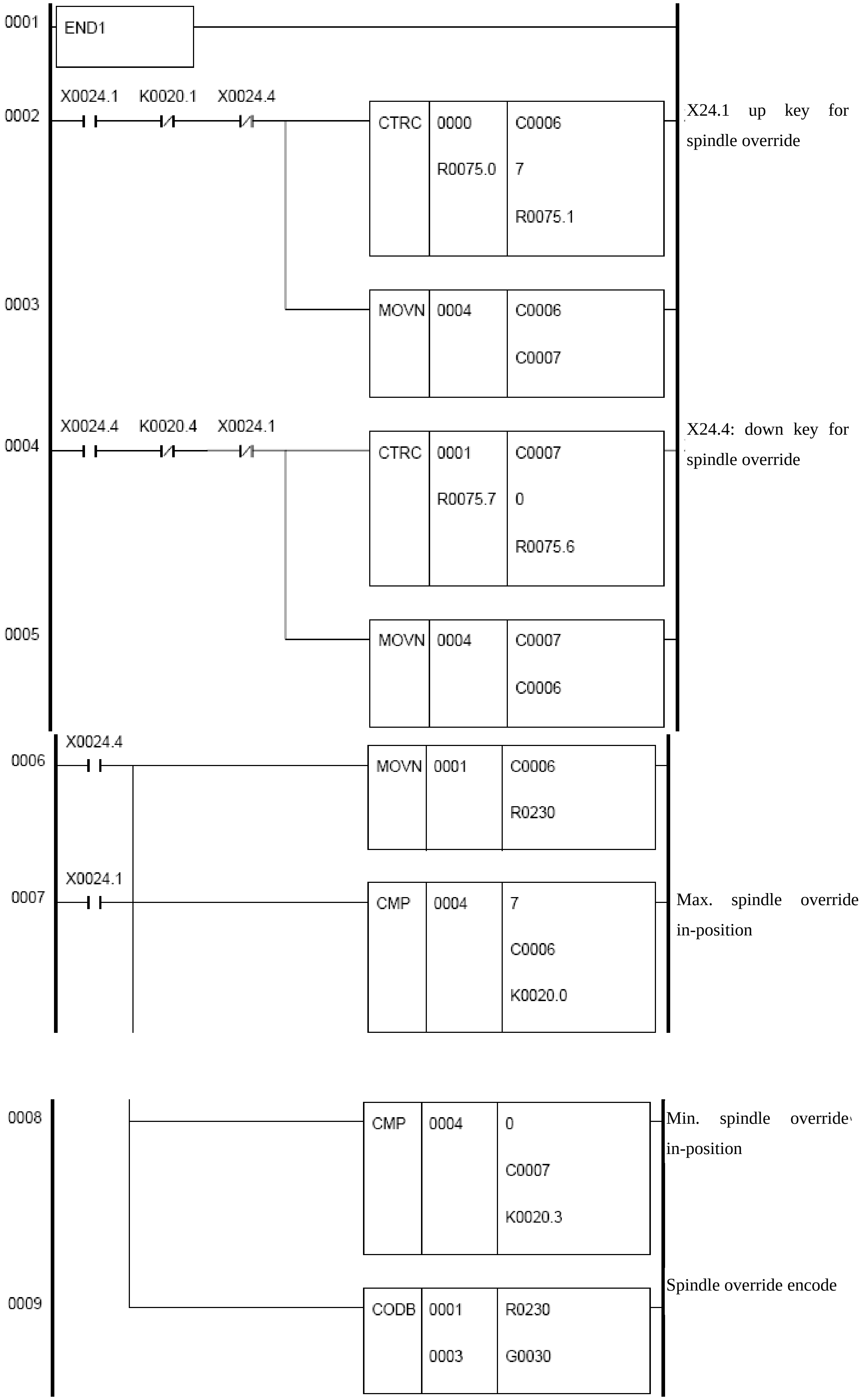
SOV7～SOV0（G30.7～G30.0）	Spindle override
0000,0111	50％
0000,0110	60％
0000,0010	70％
0000,0011	80％
0000,0001	90％
0000,0000	100％
0000,0100	110％
0000,0101	120％

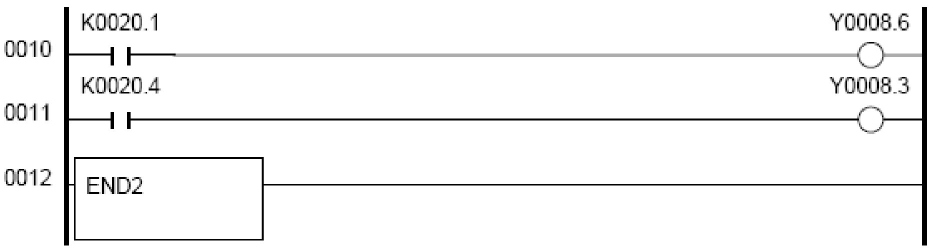
●Caution: The spindle override function is value in tapping cycle and thread cutting.

●Signal address：

G30								
	SOV07	**SOV06**	**SOV05**	**SOV04**	**SOV03**	**SOV02**	**SOV01**	**SOV00**

PLC control flow of spindle override:





Y8.6: light for max. spindle override

Y8.3: light for min. spindle override

Spindle enabling signal ENB（F001.4）

●Signal type：NC→PLC

●Signal function: it represents whether there is the spindle instruction.

●Caution: when an instruction sets the spindle speed(it is not zero) to output to the spindle, NC set F1.4 to 1; if it is 0, ENB signal is set to 0. In analog spindle mode, even if the input spindle instruction is 0(namely, analog voltage is 0V), the spindle motor rotates with low speed owing the voltage excursion of inverter. At this time, ENB signal is used for stopping spindle motor.

●Signal address：

F1				**ENB**				

Gear selection signal GR1,GR2（G28.1，G28.2）

●Signal type：PLC→CNC

● Signal function：it informs CNC of current selected gear and its operations is referred to the above-mentioned.

●Signal address：

G28						GR2	GR1	

S 12-bit code signal R01O~R12O（F036.0~F037.3）

●Signal type：NC→PLC

●Signal function: it converts instruction value of spindle speed counted by NC spindle control function into 0～4095 and outputs to PLC.

●Signal address：

F36	**R08O**	**R07O**	**R06O**	**R05O**	**R04O**	**R03O**	**R02O**	**R01O**
F37					**R12O**	**R11O**	**R10O**	**R09O**

8.2.2 spindle output control of PLC

PLC can be used for control spindle gear shifting as follows:

1：Set SIND to 1 to convert its control mode from NC to PLC.

2：Set the spindle motor speed data counted by PLC in PLC spindle control signal R01I～R12I.

When the spindle motor speed is controlled by PLC, it is influenced by any signals (such as spindle override signal) which are related to CNC spindle speed instructions or parameter settings.

Instruction selection signal of spindle motor speed SIND (G33.7)

●Signal type：PLC→NC

●Signal function: it is used for selecting CNC or PLC to control spindle motor speed.

SIND=1：control the spindle motor according to speed instruction（R01I～R12I）transmitted by PLC

SIND=0：control the spindle motor according to speed instruction transmitted by PLC. The spindle speed is output by specified by S instruction.

●Signal address：

G33	**SIND**							

S 12-bit code input signal R01I～R12I（G32.0~G33.3）

●Signal type：PLC→NC

●Signal function: it is instruction value the spindle motor speed transmitted from PLC. It specifies the spindle motor speed in binary and its speed is 0～4095.

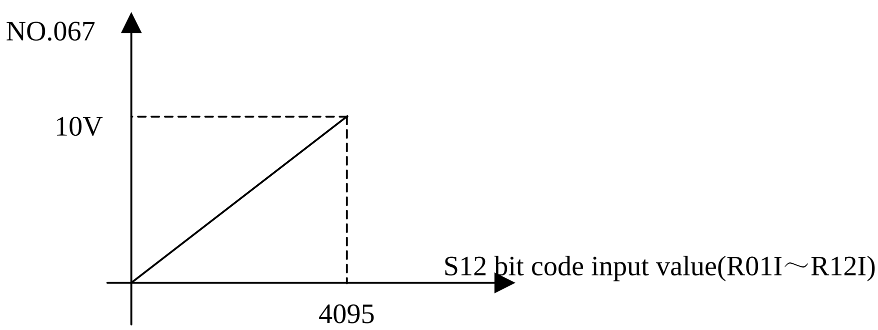
●Signal address：

G32	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G33					R12I	R11I	R10I	R09I

Instruction signal of spindle gear shifting SFTREV_L, SFTREV_H（F215, F216）

●Signal type：NC→PLC

●Signal function: the signal informs PLC of the spindle motor speed instruction in gear shifting and specifies the spindle motor speed in binary. Its speed is 0～4095. Relation between the speed and 10V is as follows（F216,F215）=NO.067×4095/10000）:



●Signal address：

F215	**SFTREV_L**
F216	**SFTREV_H**

Book 2 Function

Valid signal of automatic gear shifting AGER（F208.7）

●Signal type：NC→PLC

●Signal function: NC transmits AGER(F208.7) to PLC and PLC executes logical control based whether AGER(F208.7) is 1. When AGER(F208.7) is 1, the automatic gear shifting is valid and PLC executes the automatic gear shifting.

●Signal address：

F208	AGER							

Test gear shifting in-position signal AGIN（F208.6）in automatic gear shifting to 1, 2

●Signal type：NC→PLC

●Signal function: NC transmits AGIN（F208.6） to PLC. When AGIN（F208.6）is 1, the system automatically gear shifts to 1 or 2, and AGIN（F208.6）is 0, PLC does not test the gear shifting in-position signal.

●Signal address：

F208		AGIN						

8.3 SPINDLE JOG FUNCTION

When the spindle jog function is valid, the spindle rotates at the speed set by NO.109 for some time set by NO.108 after it is started. The function is valid for analog spindle control and executed in JOG, Handwheel/Increment, Machine Reference Point Return and Program Reference Point Return mode.

8.3.1 Spindle Jog Signal

SPHD（G200.0）

●Signal type：PLC→NC

●Signal function: when the analog spindle function is valid, PLC sets G200.0 to 1 and transmits to 1 to specify that the spindle jog function is valid. G200.0 is 1 when the spindle rotates normally.

●Signal address：

G200								SPHD

8.3.2 Spindle Jog Detection Signal

MSPHD（F199.0）：

●Signal type：NC→PLC

●Signal function: when CNC is executing the spindle jog, NC sets F199.0 to 1 and transmits to PLC to inform PLC that NC is executing the spindle jog; F199.0 is 0 when NC is not the spindle jog mode.

●Signal address:

F199								MSPHD

8.4 SPINDLE SPEED IN-POSITION SIGNAL

CNC can start the cutting feed when the spindle speed in-position SAR is input signal. Usually, when the spindle speed is the specified one, the cutting feed is started. At this time, sensor detects the spindle speed, and the detected one is transmitted to NC by PLC.

When PLC program is edited for spindle speed in-position, if instructions of spindle speed changing and cutting feed are transmitted simultaneously, CNC changes its state according to the spindle speed to confirm SAR signal to start cutting feed. To avoid the above-mentioned problem, CNC needs to delay SAR signal and delay time is set by NO.072 after S instruction and cutting feed are transmitted.

NO.175 BIT6 is set to 1 when SAR signal is used.

Spindle speed in-position signal SAR（G029.4）:

●Signal type：PLC→NC

●Signal function: when PLC sets G29.4 to 1, it transmits to NC and informs NC that the cutting feed can be started. When G29.4 is 0, the cutting feed cannot be started.

●Caution: the signal is used for informing CNC that the spindle speed is in-position. And so the signal is set to 1 after the actual spindle speed is the specified one. CNC detects SAR signal in the following states:

A: After CNC switches from rapid traverse movement mode to cutting feed, before the first feed block is distributed, and after CNC reads the feed block and runs a period of time set by NO.072, CNC starts to detect SAR.

B: After S code is commanded and before the first feed block is distributed, its dwell time is the same as the above-mentioned A.

C: When S code and the cutting feed are in the same block, and after S code is output, CNC dwell a period of time set by NO.072 and starts to detect SAR signal. If SAR signal is 1, CNC starts to cutting feed.

●Signal address：

G29				**SAR**				

8.5 RIGID TAPPING

Rigid tapping is defined that the tapping axis and the spindle are executed synchronously in general tapping fixed cycle. CNC needs to test directional signal of spindle rotating in rigid tapping to ensure that the cutting feed direction and machining process are executed normally.

Execution process: spindle rotating →tool infeed to tapping in Z direction→transmit M05 to spindle →wait for spindle complete stop →transmit instruction for spindle counterclockwise rotation →tool retraction to starting point in Z direction →spindle stop. To execute the rigid tapping, the ladder must be edited to inform CNC of rotating direction of external spindle.

Book 2 Function

SRVB（G74.4）, SFRB（G74.5）:

●Signal type：PLC→NC

●Signal function: PLC sets G74.4, G74.5 in rigid tapping, transmits to NC and inform it that the spindle rotates clockwise or counterclockwise to start cutting feed; when the spindle rotates clockwise, SRVB=1; when it rotates counterclockwise, SFRB=1; when it stops, SRVB, SFRB are set to 0.

●Caution: PLC must set SRVB, SFRB and informs CNC of current spindle rotating direction in rigid tapping.

●Signal address：

G74			**SFRB**	**SRVB**				

Chapter 9　TOOL FUNCTION

When the T code or HDT signal is specified, NC compares the required tool number with the current tool number NT00～NT31, and it does not execute the tool change if the tool numbers are not consistent with each other, which generates a code signal and general signal of the required tool number, and the machine selects tool according to the generated signal.
The tool change can be executed with T instruction in Auto/MDI/Manual mode and the sequential tool change is executed by HDT signal.

Current tool number NT00～NT31（G201～G204）

●Signal type：PLC→NC

●Signal function: After PLC has tested the current tool selection signal, it sets NT00～NT31（G201～G204） to the corresponding value, transmits to NC and informs NC of current tool number. These signals are to represent tool numbers with binary code.

●Signal address

G201	NT07	NT06	NT05	NT04	NT03	NT02	NT01	NT00
G202	NT15	NT14	NT13	NT12	NT11	NT10	NT09	NT08
G203	NT23	NT22	NT21	NT20	NT19	NT18	NT17	NT16
G204	NT31	NT30	NT29	NT28	NT27	NT26	NT25	NT24

9.1 TOOL CHANGE with T INSTRUCTION

User can specify T instruction to tool change in Auto/MDI mode; NC transmits tool selection number and general signal specified by T instruction and then waits for PLC to execute tool change after it explains T instruction. Refer to ***Chapter 7*** about tool change.

9.2 MANUALLY SEQUENTIAL TOOL CHANGE

The current tool number adds 1 as the next tool to execute tool change in manual sequential tool change. If the current tool number adding 1 is more than the total tool selections set by NO.084, the required next tool number is 1.

Execution process of CNC manual sequential tool change is the same that of tool change with T instruction; NC transmits the tool selection number and general signal of the next tool and waits for PLC executing tool change. CNC transmits the tool number which the current one adds 1 in manual sequential tool change, but one is specified by T instruction in tool change with T instruction.

Manually sequential tool change HDT（G44.7）：

●Signal type：PLC→NC

●Signal function: PLC sets G44.7 to 1 and transmits to NC; NC will execute tool change sequentially according to current tool number.

●Signal address：

G44		HDT							

Chapter 10　PROGRAMMING INSTRUCTION

In programming, although subprogram is useful for repetitively executing the same operations, macro program is useful for editing general program because it can use variable, arithmetical operation, logic operation and condition branch. A part program can call a user macro program like calling a subprogram as follows:

```
○0001（main program name）;
N10 G50 X100 Z100;
N20 G00 U50 F100;
N30 G01 U0.8;
N40 M61;
N50 G0 X100 Z100;
N60 M99;
```

```
Subprogram（custom M61）
○9064;
N10 G65 P#1104 Q1;
G65 H82 P20 Q#1004 R1;
,,,,,,,,,,,,,,,,,,,,,,
G65 H01 P#1100 Q0;
M99 P50;
```

Some function programmed by macro program is usually used for module. Programs can be edited by data variable and can be compiled by macro program to form the machining technology.

Book 2 Function

10.1 INPUT SIGNALS of USER MACRO PROGRAM

UI000～UI015（G054, G055）:

●Signal type：PLC→CNC

●Signal function: They are used for CNC variable of user macro program read/write, and for interface signal between user macro program and PLC instead of offering any functions for control units. Their corresponding variables are as follows:

G54	**UI007**	**UI006**	**UI005**	**UI004**	**UI003**	**UI002**	**UI001**	**UI000**
Variable	#1007	#1006	#1005	#1004	#1003	#1002	#1001	#1000

G55	**UI015**	**UI014**	**UI013**	**UI012**	**UI011**	**UI010**	**UI009**	**UI008**
Variable	#1015	#1014	#1013	#1012	#1011	#1010	#1009	#1008

●Signal address：

#1032	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0
	UI007	**UI006**	**UI005**	**UI004**	**UI003**	**UI002**	**UI001**	**UI000**
	BIT15	BIT14	BIT13	BIT12	BIT11	BIT10	BIT9	BIT8
	UI015	**UI014**	**UI013**	**UI012**	**UI011**	**UI010**	**UI009**	**UI008**

10.2 OUTPUT SIGNALS of USER MACRO PROGRAM

UO000～UO015（F054～F055）, UO100～UO131（F056～F059）:

- Signal type：CNC→PLC
- Signal function: They are used for CNC variable of user macro program read/write and for interface signal between user macro program and PLC instead of offering any functions for control units. Their corresponding variables are as follows:

F54								
F54	**UO007**	**UO006**	**UO005**	**UO004**	**UO003**	**UO002**	**UO001**	**UO000**
Variable	#1107	#1106	#1105	#1104	#1103	#1102	#1101	#1100

F55	**UO015**	**UO014**	**UO013**	**UO012**	**UO011**	**UO010**	**UO009**	**UO008**
Variable	#1115	#1114	#1113	#1112	#1111	#1110	#1109	#1108

F56	**UO107**	**UO106**	**UO105**	**UO104**	**UO103**	**UO102**	**UO101**	**UO100**
F57	**UO115**	**UO114**	**UO113**	**UO112**	**UO111**	**UO110**	**UO109**	**UO108**
F58	**UO123**	**UO122**	**UO121**	**UO120**	**UO119**	**UO118**	**UO117**	**UO116**
F59	**UO131**	**UO130**	**UO129**	**UO128**	**UO127**	**UO126**	**UO125**	**UO124**

- Signal address：

#1132	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0
	UO007	**UO006**	**UO005**	**UO004**	**UO003**	**UO002**	**UO001**	**UO000**
	BIT15	BIT14	BIT13	BIT12	BIT11	BIT10	BIT9	BIT8
	UO015	**UO014**	**UO013**	**UO012**	**UO011**	**UO010**	**UO009**	**UO008**

#1133	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0
	UO107	**UO106**	**UO105**	**UO104**	**UO103**	**UO102**	**UO101**	**UO100**
	BIT15	BIT14	BIT13	BIT12	BIT11	BIT10	BIT9	BIT8
	UO115	**UO114**	**UO113**	**UO112**	**UO111**	**UO110**	**UO109**	**UO108**
	BIT23	BIT22	BIT21	BIT20	BIT19	BIT18	BIT17	BIT16
	UO123	**UO122**	**UO121**	**UO120**	**UO119**	**UO118**	**UO117**	**UO116**
	BIT31	BIT30	BIT29	BIT28	BIT27	BIT26	BIT25	BIT24
	UO131	**UO130**	**UO129**	**UO128**	**UO127**	**UO126**	**UO125**	**UO124**

Chapter 11 DISPLAY/SET/EDIT

11.1 SOFT OPERATOR PANEL

Soft operator panel function is to replace some control switches on the machine operator panel with soft switches. Connect or disconnect with soft switches by using keys on MDI panel.
CNC informs PLC that states of all soft switches are indicated by their output signals. For example, soft switch SBKO is set to 1, and CNC is not in Single Block mode. Only when PLC set SBK to 1, CNC can select Single Block mode.

Auxiliary function locked soft key AFLO（F75.1）
Program jump soft key BDTO（F75.2）
Single Block soft key SBKO（F75.3）
Machine locked soft key MLKO（F75.4）
Dry run soft key DRNO（F75.5）
OUTO soft key（F197.1）

●Signal type：NC→PLC
●Signal function: NC sets the above-mentioned F signals, and transmits them to PLC; PLC sets the corresponding F signals and then transmits them to NC to select their corresponding functions.
●Signal address：

F75			**DRNO**	**MLKO**	**SBKO**	**BDTO**	**AFLO**	
F197							**OUTO**	

Auxiliary function signal AFL（G5.6）
Program jump signal BDT（G44.0）
Single Block signal SBK（G46.1）
Machine locked signal MLK（G44.1）
Dry run signal DRN（G46.7）
●Signal type：PLC→NC
●Signal function: PLC sets the above-mentioned G signals, and transmits to NC which will define their corresponding functions.
●Signal address

G5		**AFL**						
G44							**MLK**	**BDT**
G46	**DRN**						**SBK**	

11.2 MEMORY PROTECTION KEY

The memory protection key is used for protecting part programs, offset values, and parameters from being modified by mistake.
Memory protection signal KEY1（G46.3）

Book 2 Function

●Signal type：PLC→NC

●Signal function: Set whether memory content can be modified.

KEY1＝1：inputs of programs, offset values and parameters are valid;

KEY1＝0：inputs of programs, offset values and parameters are invalid.

●Signal address

G46					**KEY1**			

Chapter 12　FUNCTION PARAMETER SIGNAL

The function parameter signals are defined that CNC converts function parameters related to PLC to F signals to output it. PLC optionally executes a concrete function set by CNC parameter according to these function parameter signals.

12.1 PARAMETER SIGNALS RELATED to SPINDLE

CNC informs PLC of signals related to spindle, including spindle control mode, automatic gear shifting, rotation enabling test.

12.1.1 Spindle Control Mode Signal

SIMSPL（F200.4）:

●Signal type: NC→PLC, refer to ***Chapter 8*** in the book

Book 2 Function

12.1.2 Spindle Rotation Enabling Signal

SPEN（F208.3）:

●Signal type: NC→PLC

●Signal function: set whether the spindle rotation enabling signal test is valid.

SPEN＝0: the spindle rotation enabling signal test is invalid.

SPEN＝1:and AGIN（F208.6）＝0: the detection for spindle rotation enabling signal is valid.

●Signal address:

F208								
					SPEN			

12.1.3 Automatic Gear Shifting Function Signal

AGER（F208.7）:

●Signal type: NC→PLC, refer to 8.2.2

12.1.4 Checking Gear Shifting In-position Signal When Automatic Shifting to Gear 1 or 2

AGIN（F208.6）

●Signal type: NC→PLC, refer to 8.2.2

12.1.5 Gear Shifting In-position Signal/Level Selection of Spindle Enabling Signal

AGIM（F208.5）:

●Signal type: NC→PLC

●Signal function: It informs PLC of valid levels of gear shifting in-position signal and spindle enabling signal.

AGIM＝0: high level of spindle gear shifting in-position signal is valid; low level of its enabling signal is valid;

AGIM＝1: low level of spindle gear shifting in-position signal is valid; high level of its enabling signal is valid.

●Signal address：

F208			AGIM					

12.1.6 Selection Signal of FS1～S4 Output

SUOS（F211.0）:

●Signal type: NC→PLC

●Signal function: it informs PLC of S1～S4 output selection when the system is in spindle gear shifting mode.

SUOS=0: S1～S4 output is valid in gear shifting spindle mode;

SUOS=1: S1, S2 output is valid and S3, S4 output is invalid in spindle gear shifting mode.

●Signal address：

F211								SUOS

12.2 PARAMETER SIGNALS for REFERENCE POINT RETURN

These parameter signals determine whether a valid level of deceleration signal and the reference point return keys are self-hold when CNC executes machine reference point return and program reference point return.

12.2.1 Selection Signal of Valid Level of Deceleration Signal

DECI（F201.5）:

●Signal type: NC→PLC

●Signal function: the signal informs PLC of valid level of current deceleration signal.

DECI＝0: low level of deceleration signal is valid;

DECI＝1: high level of deceleration signal is valid.

●Signal address：

F201			DECI					

12.2.2 Self-hold Signal of Motion Key in Reference Point Return

ZNIK（F205.2）

●Signal type: NC→PLC

●Signal function: The signal informs whether the motion key is self-hold when the machine reference point return or the program reference point is executed.

ZNIK＝0: the motion key is not self-hold when the reference point return is executed;

ZNIK＝1: t he motion key is self-hold until the reference point return is executed completely.

●Signal address:

F205						ZNIK		

12.3 PARAMETER SIGNALS RELATED to TOOL

CNC inform PLC of signals related to tool control function, including total tool selections, valid level of tool selection, valid level of toolpost clamping.

12.3.1 Total Tool Selection Signal

TMAX（F207）

●Signal type: NC→PLC

●Signal function: the signal informs PLC of total tool selections on the toolpost. The tool selection is represented with binary code. For example, the total tool selection is 4, TMAX: 00000100.

●Signal address:

F207								

12.3.2 Level Selection Signal for Tool Selection Signal

TSGN（F205.1）

●Signal type: NC→PLC

●Signal function: the signal informs PLC of the valid level of current tool selection signal.

TSGN＝0: high level of tool selection signal is valid;

TSGN=1: low level of tool selection signal is valid;

●Signal address:

F205							TSGN	

12.3.3 Level Selection Signal of Toolpost Clamping

TCPS（F205.0）:

●Signal type: NC→PLC

●Signal function: the signal informs PLC of the valid level of current tool selection signal.

TSGN＝0: low level of toolpost clamping signal is valid or the system does not test the toolpost clamping signal;

TSGN=1: high level of toolpost clamping signal is valid.

●Signal address:

F205								TCPS

12.3.4 Selection Signal of Tool Change Mode

PB5（F223.0）:

●Signal type: NC→PLC

●Signal function: it informs PLC that CNC selects tool change mode A or B (refer to ***GSK980TD Turning Machine CNC-Connection***)

Book 2 Function

PB5=1：select tool change mode A

PB5=0：select tool change mode B

●Signal address：

F223								PB5

12.3.5 Selection Signal of In-position Test

PB6（F223.2）：

●Signal type: NC→PLC

●Signal function: the signal informs PLC whether CNC tests the tool selection signal after the tool change is completed.

PB6=1：test the tool selection signal;

PB6=0：do not test the tool selection signal.

●Signal address：

F223						PB6		

12.4 PARAMETER SIGNAL RELATED to CHUCK FUNCTION

CNC inform PLC of signals related to the chuck control function, including valid signal of chuck function, internal/external chuck, chuck in-position test and valid level of chuck in-position signal and so on.

12.4.1 Valid Signal of Chuck Function

SLQP（F208.0）：

●Signal type: NC→PLC

●Signal function: it informs PLC whether the current chuck function is valid.

SLQP=0: the chuck function is invalid;

SLQP=1: the chuck function is valid.

●Signal address：

F208								SLQP

12.4.2 Valid Signal of Chuck Clamping Test Function

SLSP（F208.1）：

●Signal type: NC→PLC

●Signal function: it informs PLC whether the chuck clamping signal is tested.

SLSP=0: test whether the chuck is clamping, otherwise the spindle cannot be started;

SLSP=1: do not test whether the chuck is clamping.

●Signal address：

F208							SLSP	

12.4.3 Selection Signal of Chuck Mode

PB1（F209.0）

● Signal type: NC→PLC

● Signal function: it informs PLC that the current machine is in the internal chuck mode or the external chuck mode.

PB1=0: internal chuck mode;

PB1=1: external chuck mode.

● Signal address:

F209								PB1

12.4.4 Valid Signal of Chuck In-position Signal Test Function

PB2（F209.2）

● Signal type: NC→PLC

● Signal function: it informs PLC whether the chuck in-position signal is tested.

PB2=0: do not test the chuck in-position signal;

PB2=1: test the chuck in-position signal.

● Signal address

F209						PB2		

12.5 PARAMETER SIGNALS of PRESSURE LOW TEST FUNCTION

CNC informs PLC of signals related to pressure low test function, including valid signal of pressure low test function, valid level selection signal of pressure low input signal.

12.5.1 Valid Signal of Pressure Low Test Function

PB3（F209.4）:

● Signal type: NC→PLC

● Signal function: it informs PLC whether the current pressure low test function is valid.

PB3=0: the pressure low test function is invalid;

PB3=1: the pressure low test function is valid

● Signal address

F209				PB3				

12.5.2 Valid Level Selection Signal of Pressure Low Input Signal

SPB3（F209.5）:

● Signal type: NC→PLC

● Signal function: it informs PLC of valid level of current pressure low.

SPB3=0: CNC alarms when the high level of pressure low input signal is valid;

SPB3=1: CNC alarms when the low level of pressure low input signal is valid.

● Signal address:

F209			SPB3					

12.6 PARAMETER SIGNALS of SAFEGUARD TEST FUNCTION

CNC inform PLC of signals related to safeguard test function, including valid signal of safeguard test function, selection signal of valid level of safeguard input signal.

12.6.1 Valid Signal of Safeguard Test Function

PB4（F209.6）

●Signal type: NC→PLC

●Signal function: it informs PLC whether the current safeguard test function is valid.

PB4＝0: the safeguard test function is invalid;

PB4=1: the safeguard test function is valid.

●Signal address：

F209

	PB4						

12.6.2 Selection Signal of Valid Level of Safeguard Input Signal

SPB4（F209.7）

●Signal type: NC→PLC

●Signal function: it informs PLC of the valid level of current safeguard input signal.

SPB4＝0: the safeguard is closed in low level;

SPB4＝1: the safeguard is closed in high level.

●Signal address：

F209

SPB4							

12.7 PARAMETER SINGALS of SHIELDING EXTERNAL SIGNALS

These signals inform PLC whether CNC shields some external input signals. Shielded external input signals include an external emergent stop, an external cycle startup, an external pause, an external override signal and so on.

12.7.1 Shielding External Feedrate Override Input Signal

SOVI（F210.0）：

●Signal type: NC→PLC

●Signal function: it informs PLC whether the external feedrate override signal is shielded.

SOVI=0: the external feedrate override signal is valid;

MES=1: the external feedrate override signal is shielded.

●Signal address：

F210

							SOVI

12.7.2 Shielding External Emergent Stop Signal

MESP（F210.3）：

● Signal type: NC→PLC

● Signal function: it informs PLC whether the external emergent stop signal is shielded.

MESP=0: the external emergent stop signal is valid;

MESP=1: the external emergent stop signal is shielded.

● Signal address：

F210					MESP			

12.7.3 Shielding External Pause Signal

MSP（F210.5）:

● Signal type: NC→PLC

● Signal function: it informs PLC whether the external pause signal is shielded.

MST=0: the external pause signal is valid;

MST=1: the external pause signal is shielded.

● Signal address：

F210			MSP					

12.7.4 Shielding External Cycle Startup Signal

MST（F210.6）:

● Signal type: NC→PLC

● Signal function: it informs PLC whether the external cycle startup signal is shielded.

MST=0: the external cycle startup input signal is valid;

MST=1: the external cycle startup input signal is shielded.

● Signal address：

F210		MST						

12.8 PARAMETER SIGNAL for KEYS on OPERATOR PANEL

These signals inform PLC that CNC defines complex usages of keys on the operator panel and usages of special definitions. Such as complex usage of OUT key and lubrication key, reset key of output interface, direction selection of motion key.

12.8.1 Signal of OUT Key Startup Programs

DCS（F201.2）:

● Signal type: NC→PLC

● Signal function: it informs PLC whether pressing OUT key starts programs in MDI mode.

DCS＝0: pressing OUT key cannot start programs;

DCS＝1: pressing OUT key can start programs.

● Signal address：

F201						DCS		

Book 2 Function

12.8.2 Complex Signal of Lubrication/Spindle Jog Key

RHDO（F214.7）:

●Signal type: NC→PLC

●Signal function: it informs PLC of functions of lubrication/jog key on the operator panel.

RHDO=0: Lubrication/Jog key as Lubrication key;

RHDO=1: Lubrication/Jog key as JOG key.

●Signal address:

F214	RHDO							

12.8.3 Selection Signal of Reset Key for Output Interface

RSJG（F204.3）:

●Signal type: NC→PLC

●Signal function: it informs whether PLC resets M03, M04, M32, M08 when the reset key is pressed.

RSJG＝0: PLC automatically closes M03, M04, M32, M08 output signal when the reset key is pressed;

RSJG=1: M03, M04, M32, M08 output signals are reserved when the reset key is pressed.

●Signal address:

F204					RSJG			

12.8.4 Direction Selection Signal for Motion Key in X Direction

XVAL（F214.0）:

●Signal type: NC→PLC

●Signal function: it defines the relationship between motion key in X direction and X-axis positive/negative direction.

XVAL＝0: up motion key in X direction corresponds to X-axis negative direction, down motion key in Z direction corresponds to Z-axis positive direction;

XVAL＝1: up motion key in X direction corresponds to X-axis positive direction, down motion key in Z direction corresponds to Z-axis negative direction.

●Signal address:

F214								XVAL

12.8.5 Direction Selection Signal for Motion Key in Z Direction

ZVAL（F214.1）:

●Signal type: NC→PLC

●Signal function: it defines the relationship between motion key in Z direction and Z-axis positive/negative direction.

ZVAL＝0: left motion key in Z direction corresponds to Z-axis negative direction, right motion key in Z direction corresponds to Z-axis positive direction;

ZVAL＝1: left motion key in Z direction corresponds to Z-axis positive direction, left motion key in Z direction corresponds to Z-axis negative direction.

●Signal address：

F214							ZVAL	

12.8.6 Handwheel/Step 0.1mm Selection Signal

SINC（F211.1）：

●Signal type: NC→PLC

●Signal function: it selects whether 0.1mm increment is valid in Handwheel/Step mode.

SINC =0: 0.1mm increment is valid in Handwheel/Step mode.

SINC =1: 0.1mm increment is invalid in Handwheel/Step mode.

●Signal address：

F211							SINC	

12.9 PARAMETER SIGNALS RELATED to TAILSTOCK FUNCTION

CNC informs PLC of signals related to tailstock control function.

Valid signal of tailstock function SLTW（F208.2）

●Signal type: NC→PLC

●Signal function: it informs PLC whether the tailstock function is valid.

SLTW=0: tailstock function is invalid;

SLTW=1: tailstock function is valid;

●Signal address：

F208						SLTW		

BOOK 3

CNC CONFIGURATION SOFTWARE

Contents

Chapter 1 OVERVIEW

1.1 GSKCC SOFTWARE

With a brief software interface, GSKCC is a configuration software of GSK-980TD, used for editing PLC ladder and part programs, setting CNC parameters, tool offset, pitch compensation.

GSKCC runs in WINDOWS98/2000/XP, editing PLC ladder, setting parameters relative to CNC, saving them, and printing PLC program by printer. Its characteristics are as follows:

- Edit component comment, block comment in ladder to contribute to easy reading programs for user
- Instruction system supports subprograms and program jumping contributed to flexible programming
- Software with user environment menus and access keys is easy to operate
- Download current engineering to CNC or transmit related configuration files from CNC by serial communication interface

Refer to *Programming of GSK980TD Turning Machine CNC PLC User Manual* about CNC PLC ladder program rules and component settings, detailing operations and usages of GSKCC software（V1.1-Bulid050901）.

System requirements:

- Operating system:　　Windows98/2000/XP
- CPU:　　　　　　　Pentium 133MHz or more
- Hard disk: over 10M
- Memory: over 32M
- Display: resolution 1024*768，16 colors
- Keyboard, mouse
- Serial communication interface

1.2 DEFINITION of PROJECT

Project is defined to be an integrated file used for managing PLC, CNC configuration and part programs. PLC includes data setting list, displaying information list, ladder programs and program version information. CNC configuration includes parameters, tool offset and pitch error compensation.

Chapter 2 MENU COMMANDS

2.1 GENERAL INTERFACE

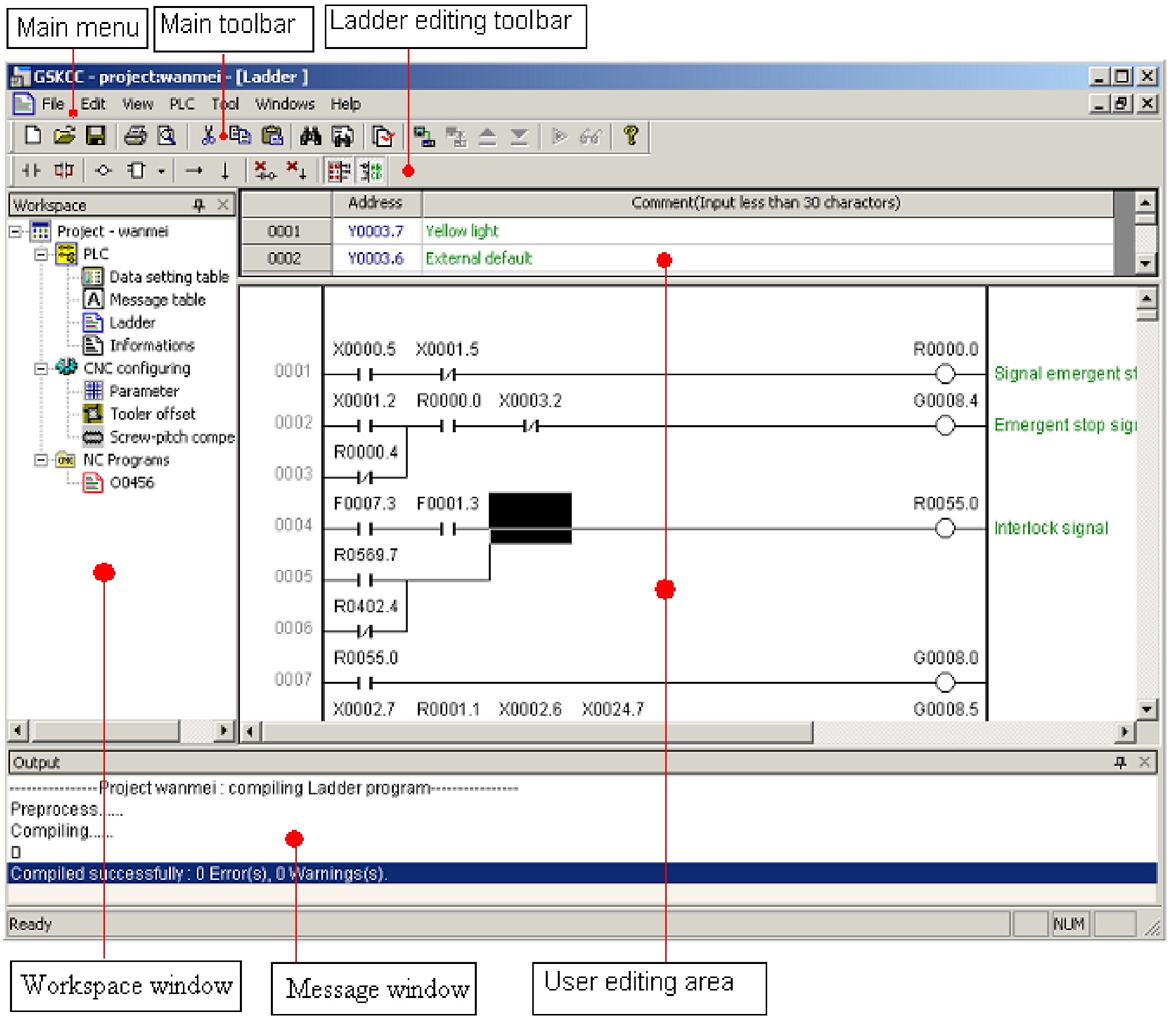


Fig. 2-1

- Main menu All software operating commands
- Main toolbar Normal operating commands
- Ladder editing toolbar Operating commands for editing ladder
- Workspace window Manage all configurations of project
- Message table Output compile message of adder and search content message
- User editing area User can select all types of view to execute many operations, including editing ladder and setting parameters

2.2 MAIN MENU COMMAND

2.2.1 [File] Menu

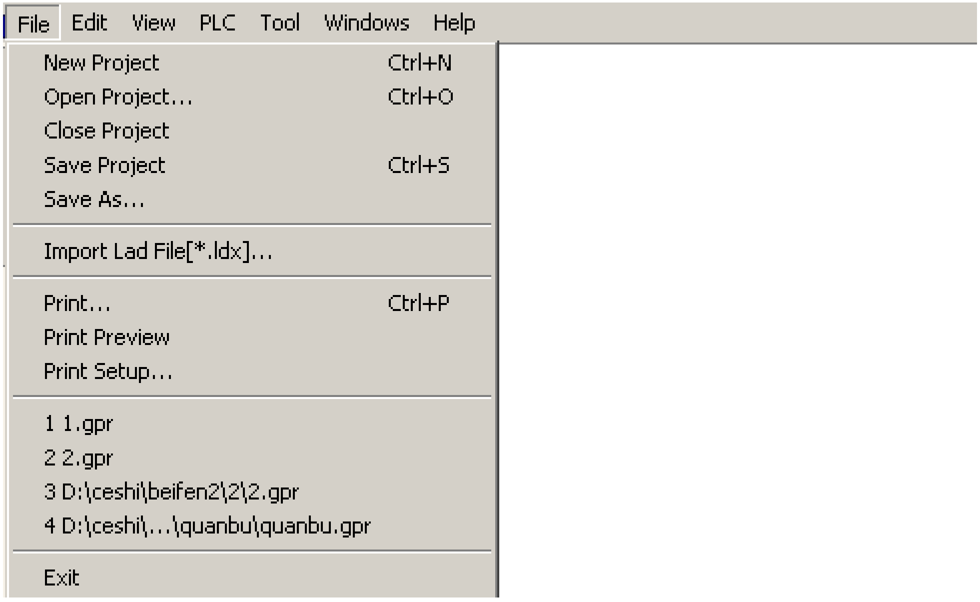
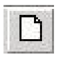


Fig. 2-2

- **[New Project]**

Create a new project by pressing access key [Ctrl+N] or mouse selecting  in the main toolbar. After executing the new project command, dialog box is displayed as follows:

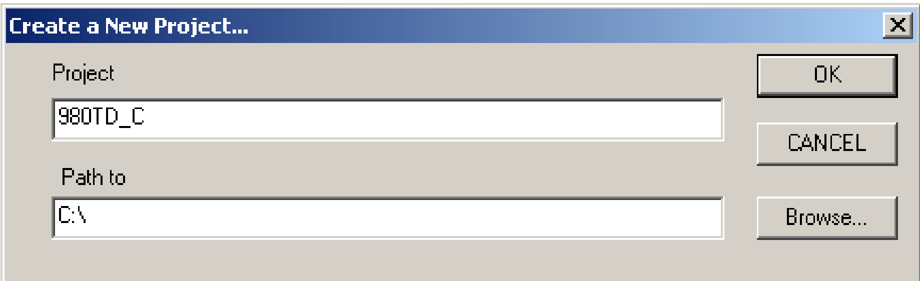


Fig. 2-3

Input one project name in the first editing box and its path to memory in the second one, or select a button to browse files at the right side, and there is a dialog box as follows, then, select one file to confirm it, and there is the selected path to memory in the second editing box:

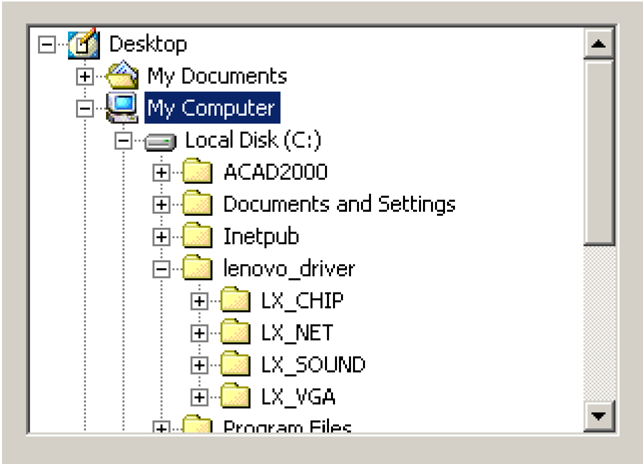


Fig. 2-4

• **[Open Project]** Open a project stored in hardware by access key [Ctrl+O] or mouse selecting in the main toolbar. After executing the open project command, there is a dialog box to open project is displayed, and then you select the project *.gpr and press [Open] to open it.

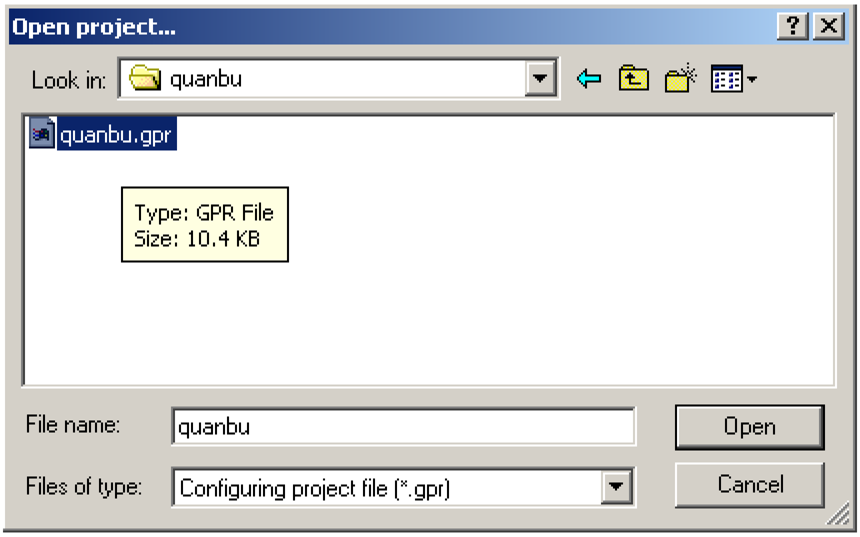


Fig. 2-5

• **[Close project]** ： Close the current open project.

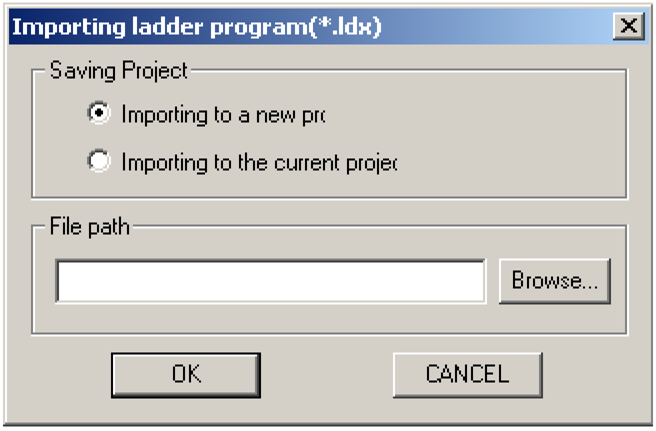
• **[Save Project]** ： Save the current open project by access key or pressing in the main toolbar.

• **[Save As]**：The current is saved as ones which is to open. After executing the command, pop-up a dialog similar to that of [New project], set its name and path to save.

• **[Import Lad File[*.ldx]]**：

Import ladder file*.ldx to project.

Execute the command and a dialog is displayed as follows:



Saving project: select **[importing to a new project]**

File path: press [Browse] and a dialog box to select files is displayed.

Press [OK] and import ladder project after selecting the saving project and file path.

• **[Print]** ： Execute the command to print the ladder when the ladder editing view is current activation one.

• **[Print Preview]**： **Firstly preview the file on screen not to print**.

• **[Last File List]** ：

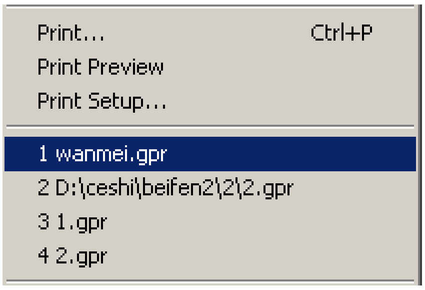


Fig. 2-6

Menu [Print Setup] lists files in which there are last 4 open files, directly press one of them to open it.

• **[Exit]**： Exit from the current application program. When the current file is not save, the program indicates if the current file is saved before exiting.

2.2.2 [Edit] Menu

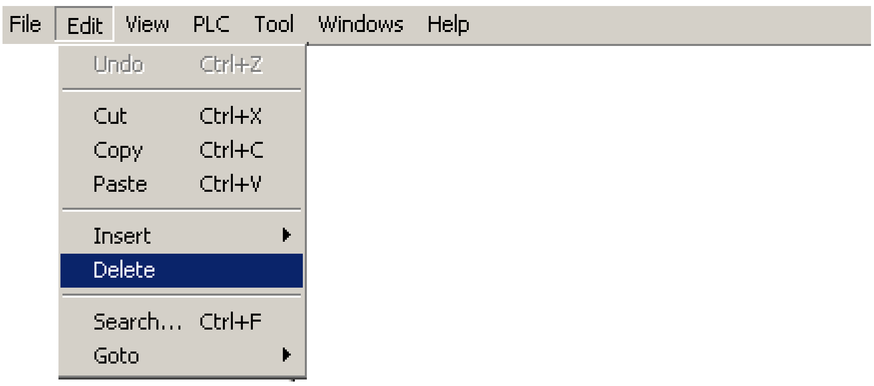


Fig. 2-7

- **[Cut]** ： Delete the selection contents to copy to clipboard by pressing access key [Ctrl+X] or in main toolbar.
- **[Copy]**： Copy the selected contents to clipboard by pressing access key [Ctrl+C] or in main toolbar.
- **[Paste]**： Paste the contents in clipboard to the selected position by pressing access key [Ctrl+C] or in main toolbar.
- **[Delete]**： Delete the selected content by access key[Delete].
- **[Insert]**： Pop-up three operating commands and insert the corresponding in the selected position.

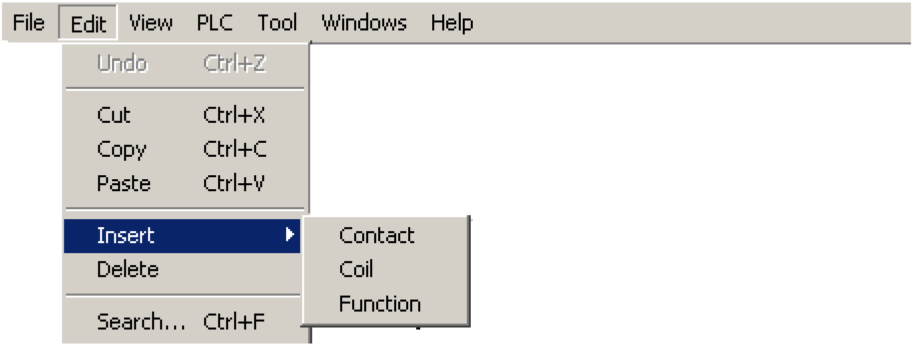


Fig. 2-8

——**[Contact]**： Insert one contact (normally-open/closed)in the selected position. Pop-up a dialog to set the contact's style and address after executing the operating command.

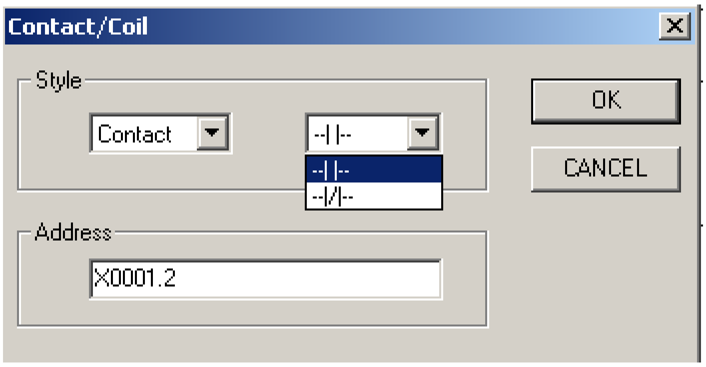


Fig. 2-9

——**[Coil]** ： Insert a output coil in the selected position. Pop-up a dialog to set the coil's style and address after executing the operating command.

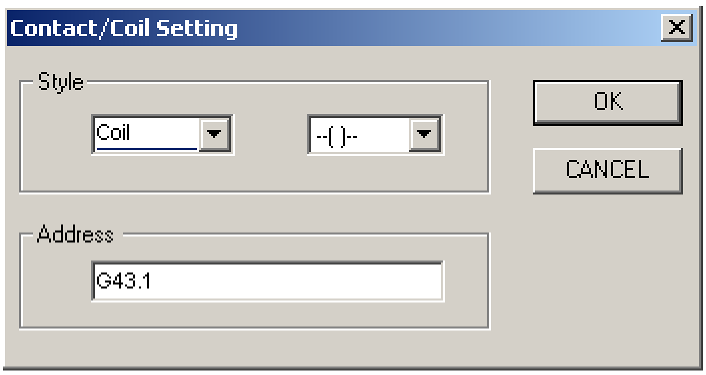


Fig. 2-10

——**[Function Instruction]:** Insert a function instruction the selected position. Pop-up a dialog on the screen after executing the operating command, and then press [OK] to pop-up a dialog of the corresponding function instruction setting after selecting the function instruction. For example, execute the operation [Edit]—[Insert]—[Function Instruction] for inserting a CODB instruction:

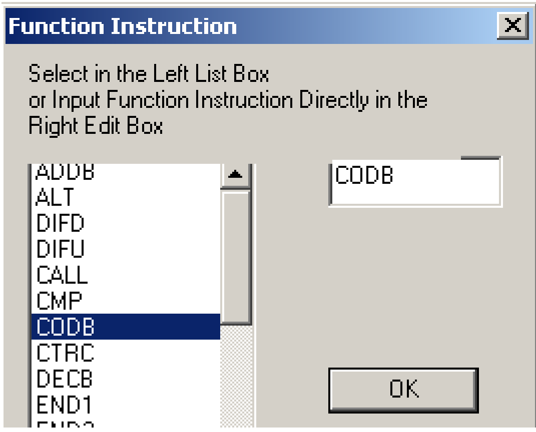


Fig. 2-11

Pop-up a setting window as Fig. 2-12 after selecting **CODB** and pressing [OK]:

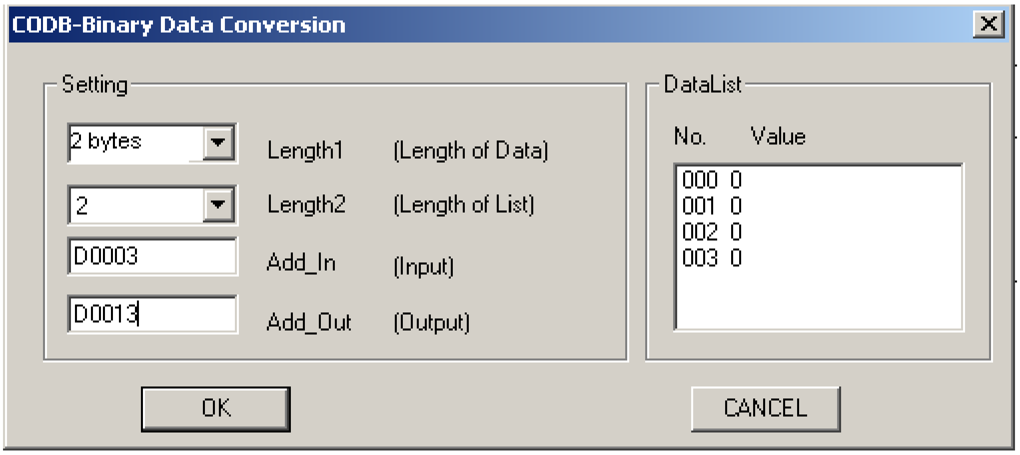


Fig. 2-12

Set the corresponding parameter in the instruction setting dialog box and press [OK].

• **[Find]:** Find the required instruction or address in the ladder program. Pop-up a dialog for finding after executing the command:

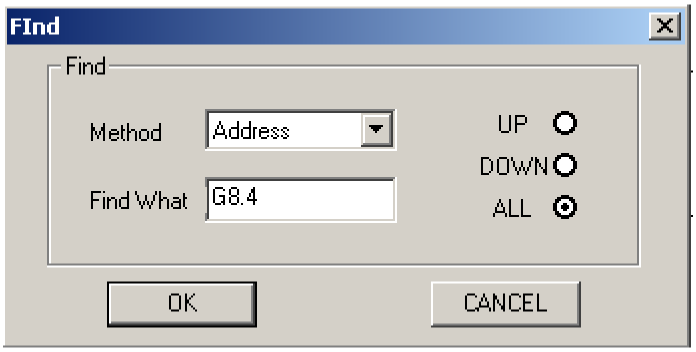


Fig. 2-13

Select [Address] or [Function Address] for finding type. Input the finding content in [Address] and press [UP], [DOWN] or [ALL] located at the right side to gradually find. The result of finding is displayed item by item in [Output] window as Fig. 2-14.

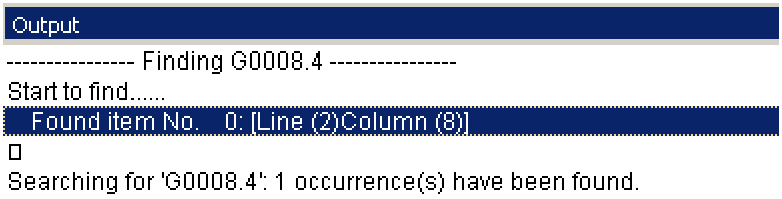


Fig. 2-14

Double click “Found item No.XX;[Line (8) Column(1)]” to scroll ladder to the corresponding position.

• **[Goto]** ： Rapidly scroll the current ladder to the required position and pop-up a menu as follows:

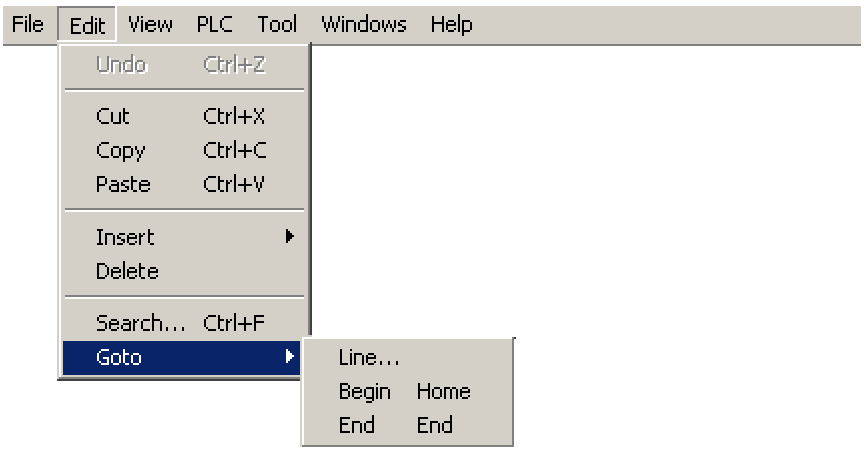


Fig. 2-15

——**[Line…]**: Rapidly scroll the ladder program to the specified line and pop-up a dialog as follows:

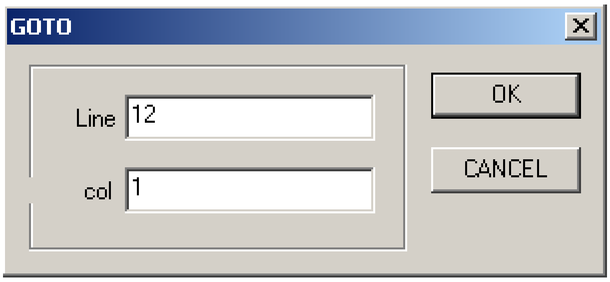


Fig. 2-16

Set line and column number of ladder , and rapidly scroll the ladder program to the specified position.

——**[Program Starting]**: Rapidly scroll the ladder program to starting position of program.

——**[End of Program]**: Rapidly scroll the ladder program to end position of program.

2.2.3 [View] Menu

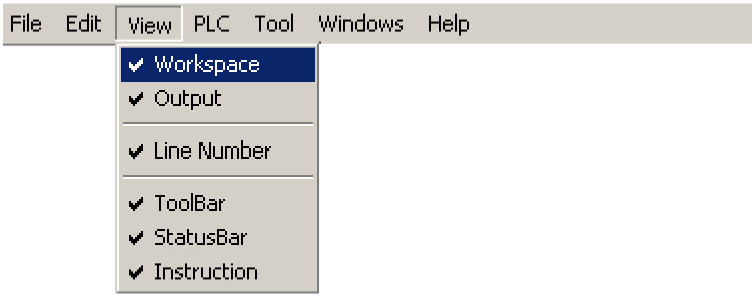


Fig. 2-17

- **[Workspace]**: display workspace window or not.
- **[Output]**: display output window or not.
- **[Line Number]**: display line number or not.
- **[Toolbar]**: display main toolbar or not.
- **[StatusBar]**: display statusBar or not
- **[Instruction]**: display instruction or not.

2.2.4 [PLC] Menu



Fig. 2-18

- **[Compile]**: Compile the current ladder program by access key F9.

2.2.5 [Tool]Menu

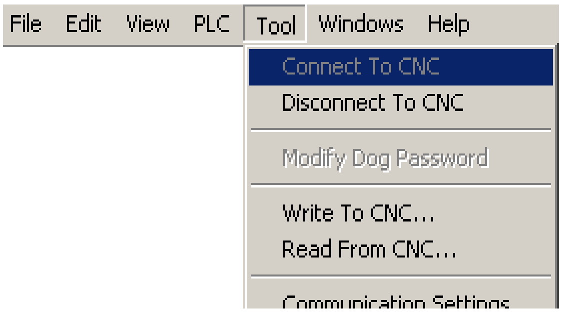


Fig. 2-19

• **[Connect To CNC]:** Execute the command and connect to CNC by serial communication interface.

• **[Disconnect To CNC]:** Disconnect to CNC and release serial communication interface.

• **[Write To CNC]:** Write the current file to CNC to store.

• **[Read From CNC]:** Read a file from CNC to PC.

•**[Communication setting]:** Set parameters of serial communication interface, including selecting serial number and baud rate.

• **[Options]:** Set the relative program options. Pop-up a dialog as follows after executing the command:

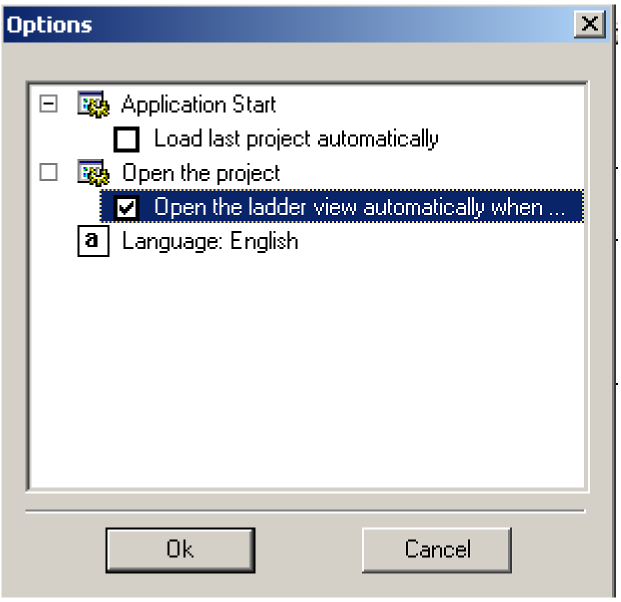


Fig. 2-20

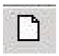
Book 3 CNC Configuration software

Chapter 3 TOOLBAR

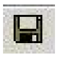
3.1 MAIN BOOLBAR



Fig. 3-1

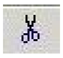
 Create new project

 Open project

 Save project

 Print

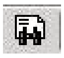
 Print Preview

 Cut

 Copy

 Paste

 Find

 Search

 Compile

 Connect to CNC

 Disconnect with CNC

 Write to CNC

 Read From CNC

3.2 TOOLBAR for EDITING LADDER

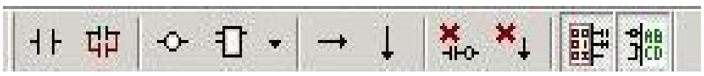


Fig. 3-2

 Insert a contact at the cursor(access key F1)

 Insert a parallel contact at the cursor

 Insert a coil at the cursor（access key F2）

 Insert a parallel contact at the cursor and press the small arrowhead to pop-up menu to select the function instruction (access key F3)

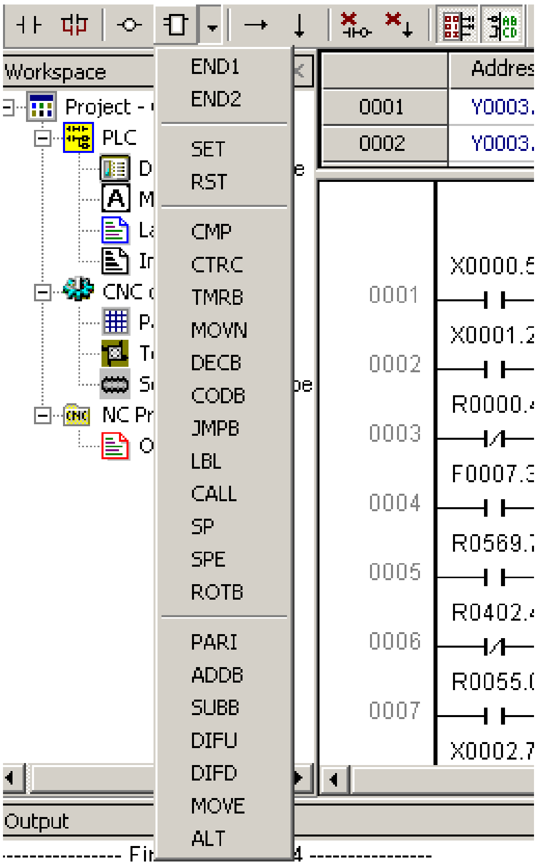
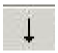
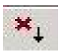


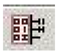
Fig. 3-3

 Insert a horizontal connecting line at the cursor (access key F4)

 Insert a vertical connecting line at the cursor

 Delete the selected element of ladder(access key Delete)

 Delete a left vertical connecting line of the selected element

 Display line number of ladder or not

 Display annotation of ladder or not

Chapter 4　SOFTWARE

4.1 WORKSPACE WINDOW

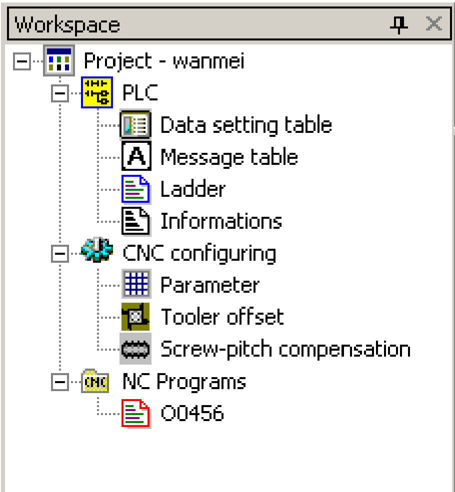


Fig. 4-1

Submenu items, including [Data setting table], [Message table], [Ladder], [Parameter] , [Tool offset], [Screw-pitch compensation], are operated by pressing right key of mouse. Pop-up a menu by pressing right key for [Ladder]

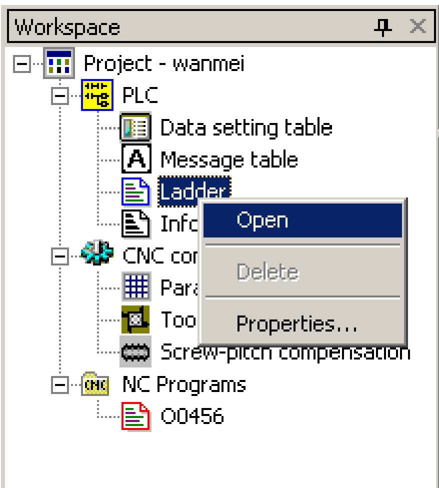


Fig. 4-2

[Open] function is the same as that of double clicking left key.

4.2 DATA SETTING TABLE

Set data in internal data register(D, DC, DT) of PLC by Data Setting Table as Fig. 4-3.

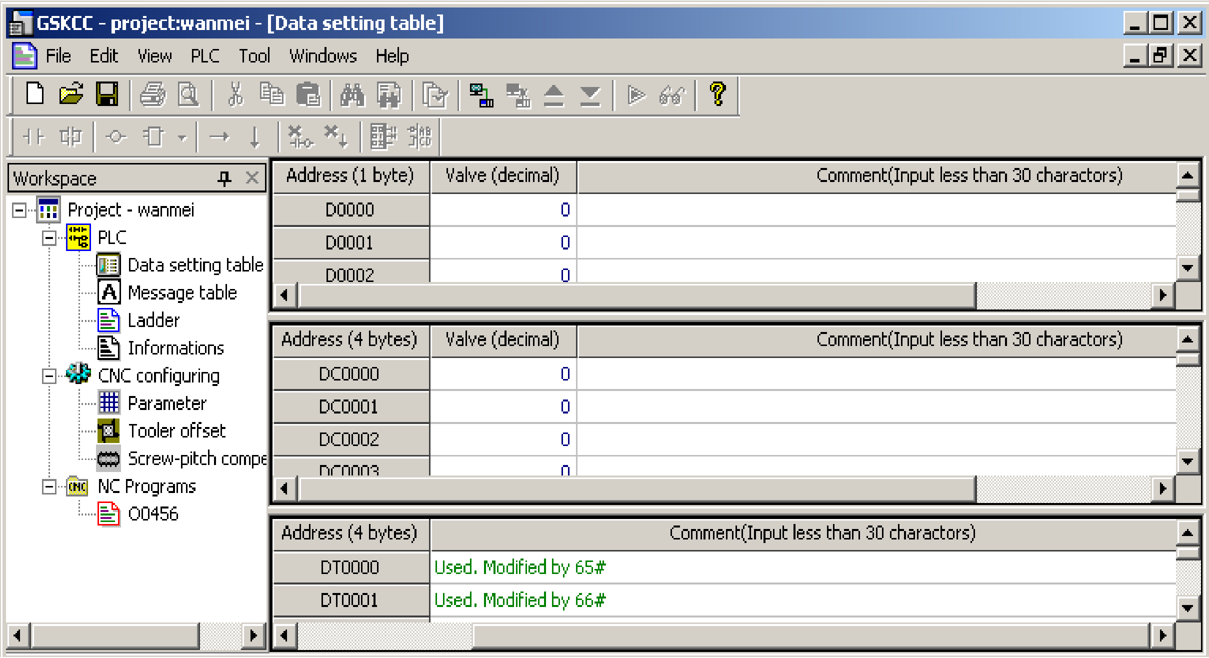


Fig. 4-3

4.3 MESSAGE TABLE

Double click [Message table] by left key or click by right key to pop a menu, and execute [Open] to open the message table as follows:

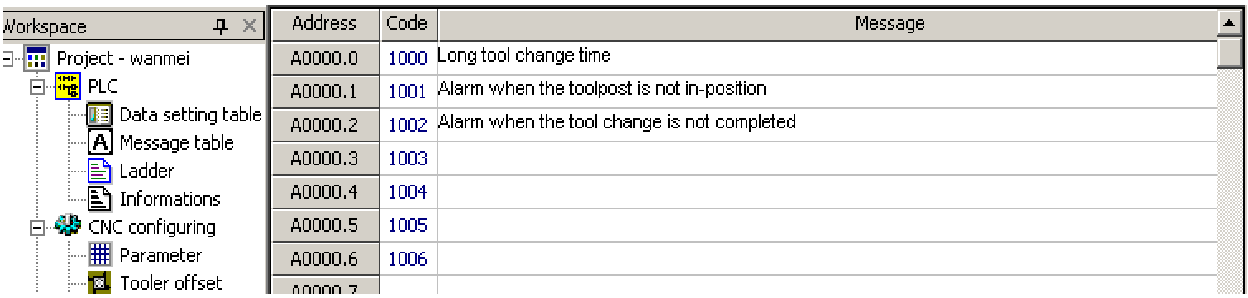
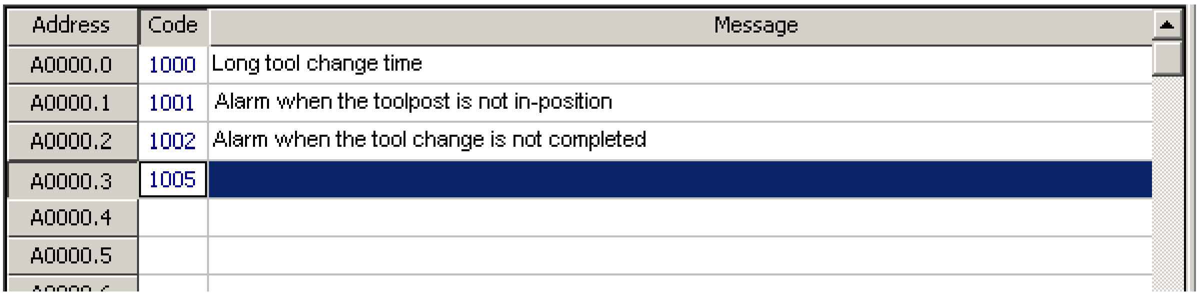


Fig. 4-4

Three lines in Message Table are as follows:

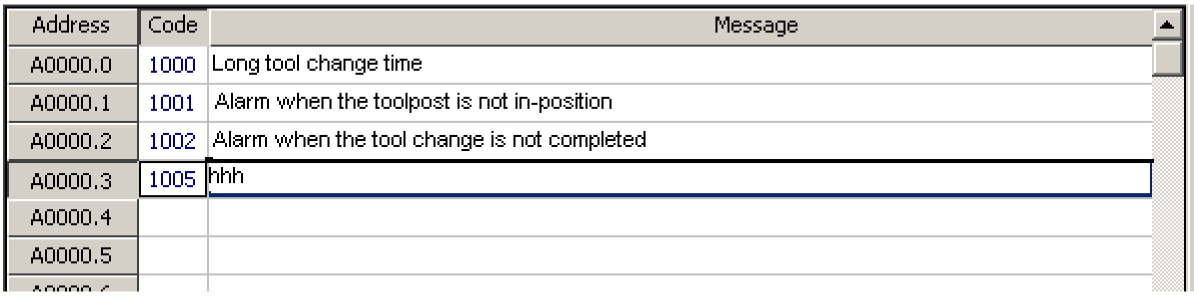
- **[A address]:** A0.0 ～ A24.7 can set 25×8=200 points for outputting coil.
- **[Code]:** Double click [Code] and there is prompt signal to input 1000 – 9999 as alarm code when CNC displays messages.



Address	Code	Message
A0000.0	1000	Long tool change time
A0000.1	1001	Alarm when the toolpost is not in-position
A0000.2	1002	Alarm when the tool change is not completed
A0000.3	1005	
A0000.4		
A0000.5		

Fig. 4-5

- **[Message]:** There is a prompt signal after inputting the coil and the input method is to double click the list box by left key, and character strings are input after there is a prompt cursor as Fig. 4-6



Address	Code	Message
A0000.0	1000	Long tool change time
A0000.1	1001	Alarm when the toolpost is not in-position
A0000.2	1002	Alarm when the tool change is not completed
A0000.3	1005	hhh
A0000.4		
A0000.5		

Fig. 4-6

4.4 LADDER

Edit the ladder program and annotation after opening its view. Refer to **Chapter 2** and **Chapter 3** about operations of each menu and toolbar.

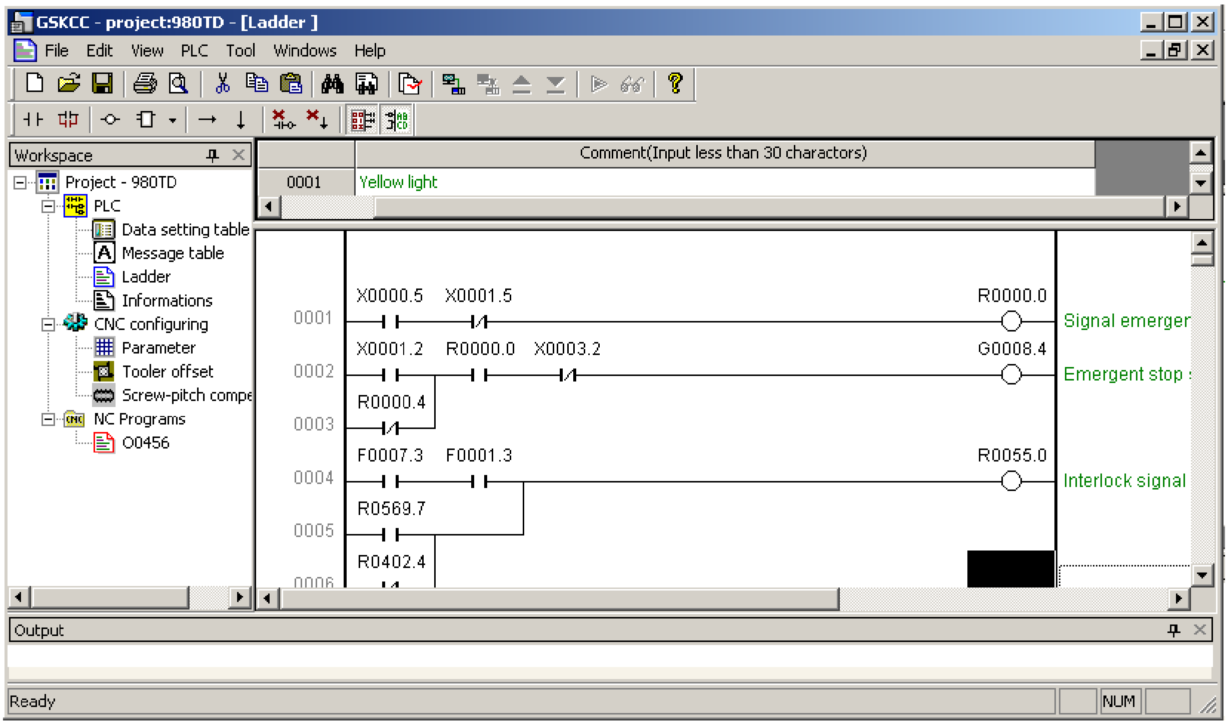


Fig. 4-7

4.5 PROGRAM VERSION

Double click the menu to pop-up its input box and input the corresponding message.

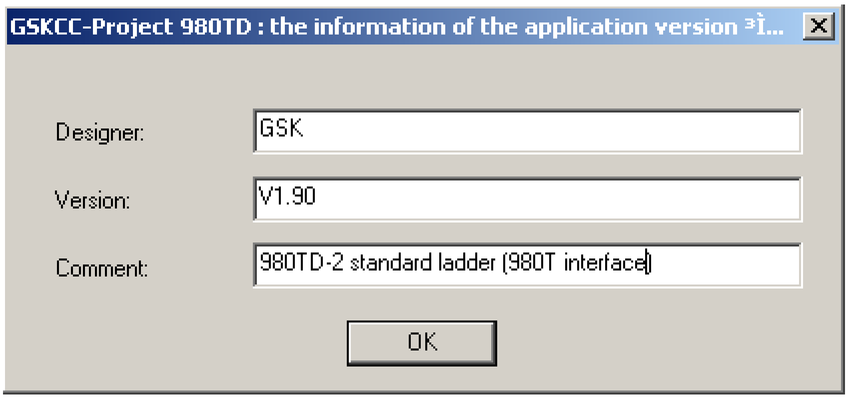


Fig. 4-8

4.6 PARAMETER

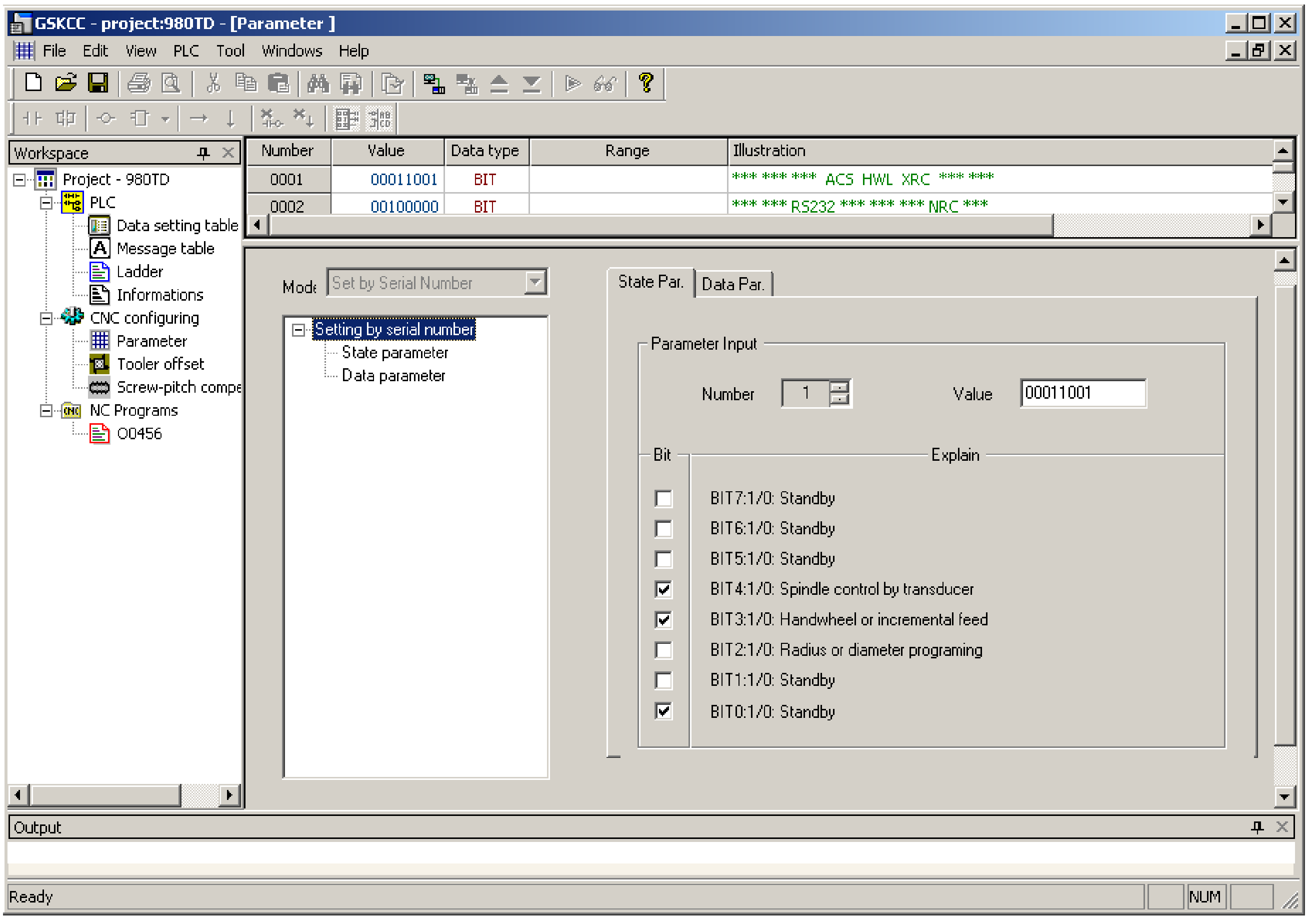


Fig. 4-9

Book 3 CNC Configuration software

4.6.1 Modifying Parameters by Sequence Number

User interface is as Fig.4-11 and the right property window is the to edit parameters when selecting sequence number to modify parameters.

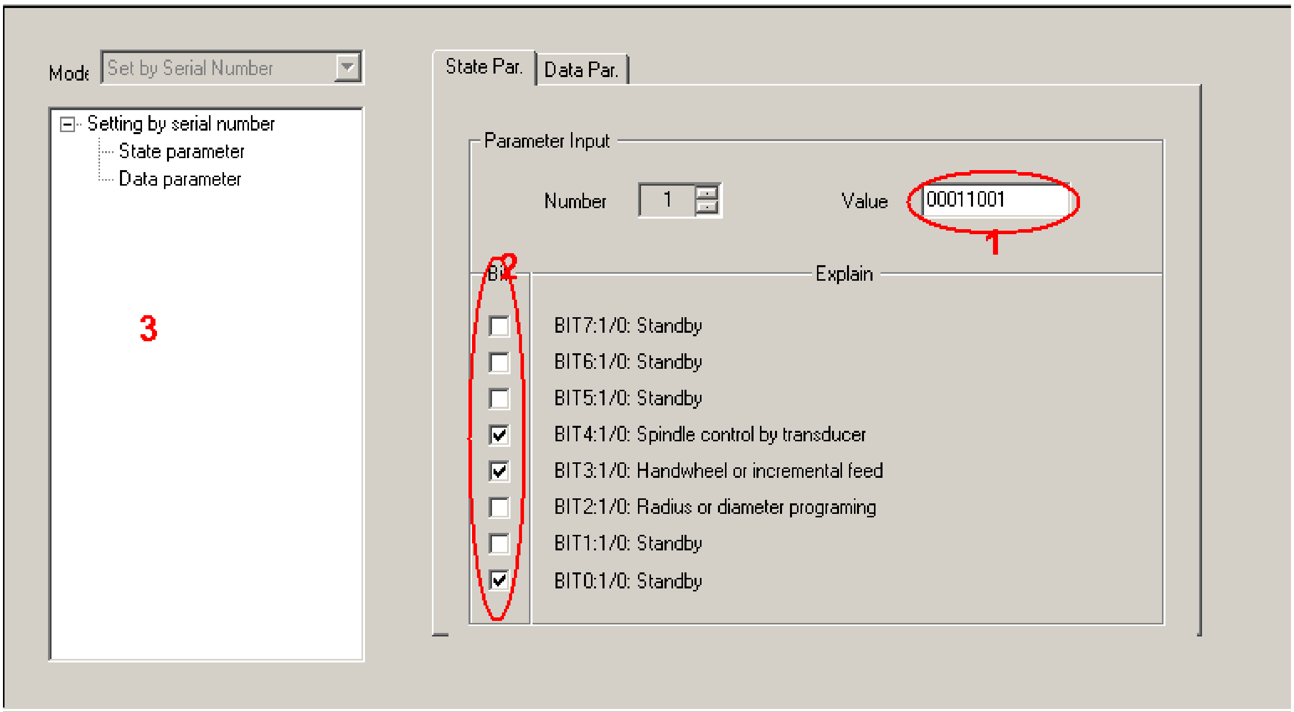


Fig. 4-11

Color cord circle in the above figure:

① Editing box for parameter;

② Choice box for bit parameter set. In the box, click the corresponding bit parameter set 1/0 by left key and the corresponding bit value will be updated in ①;

③ Menu of parameter data can be switch by right clicking the above property menu.

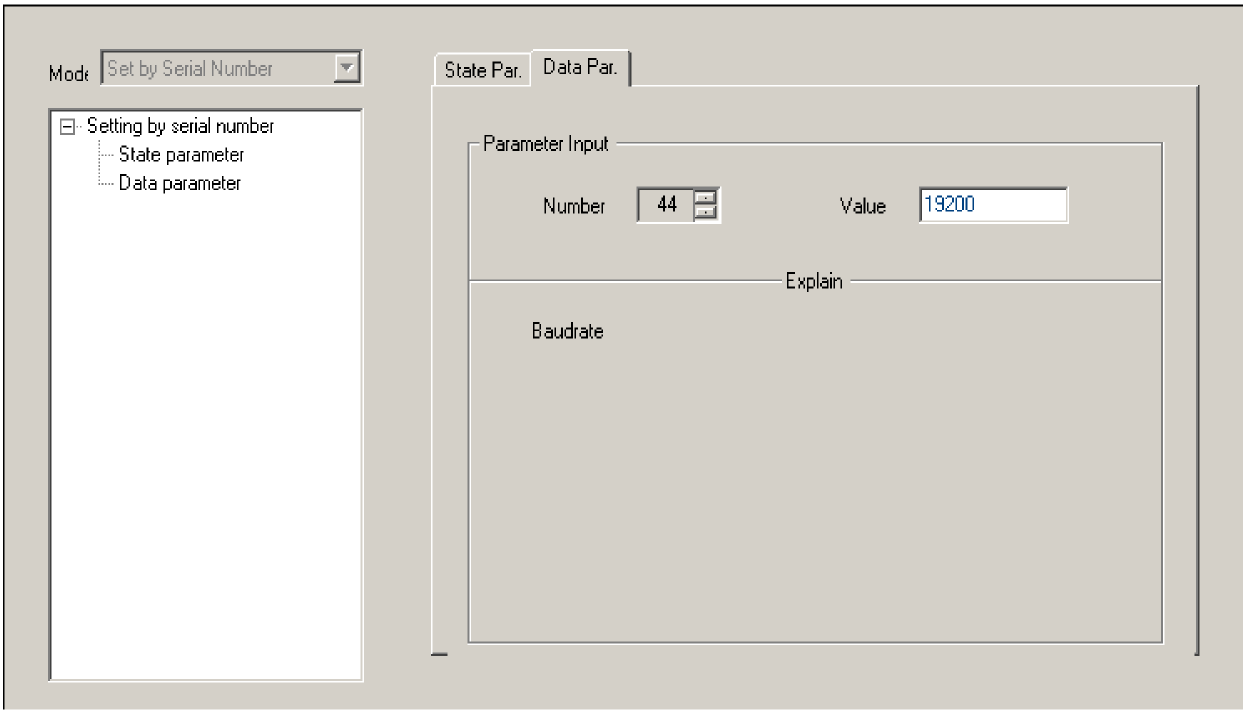


Fig. 4-12

4.7 TOOL OFFSET

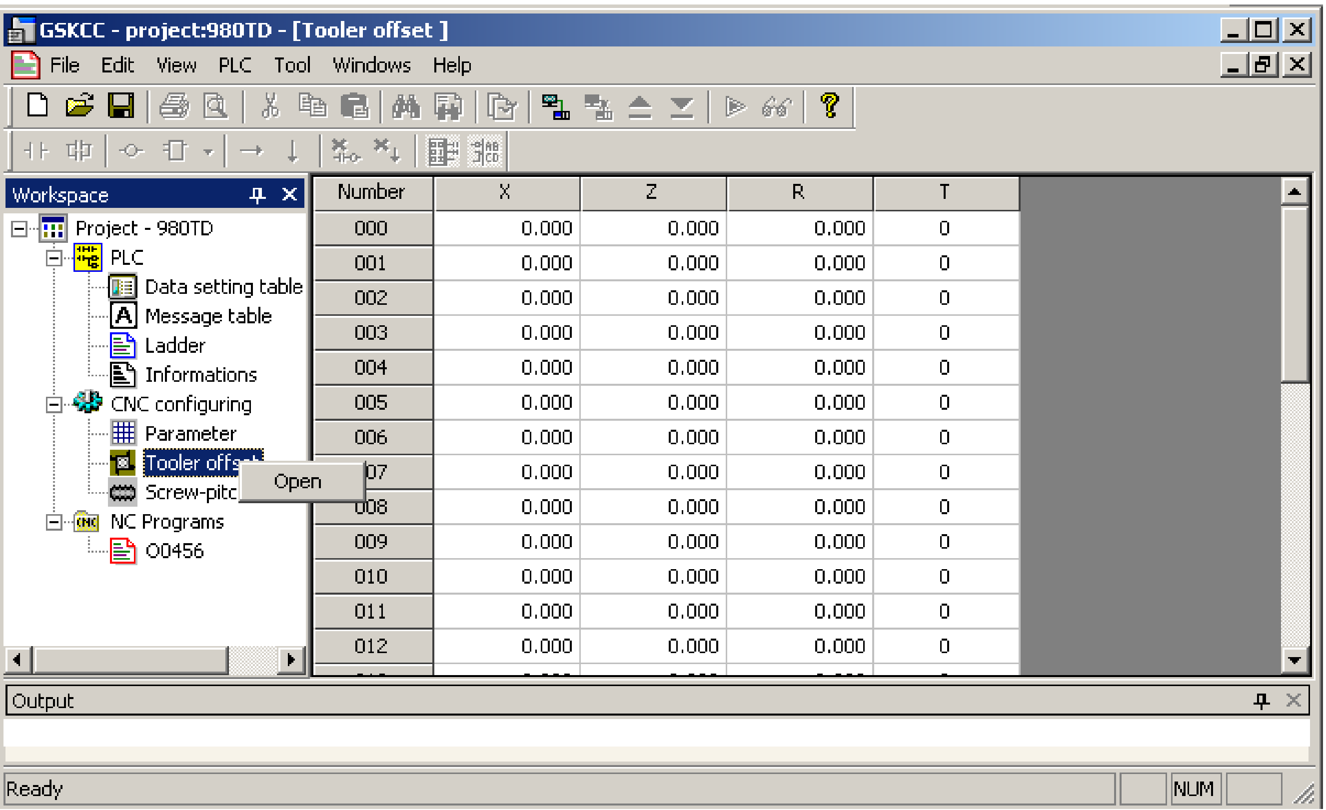


Fig. 4-14

4.8 SCREW-PITCH COMPENSATION

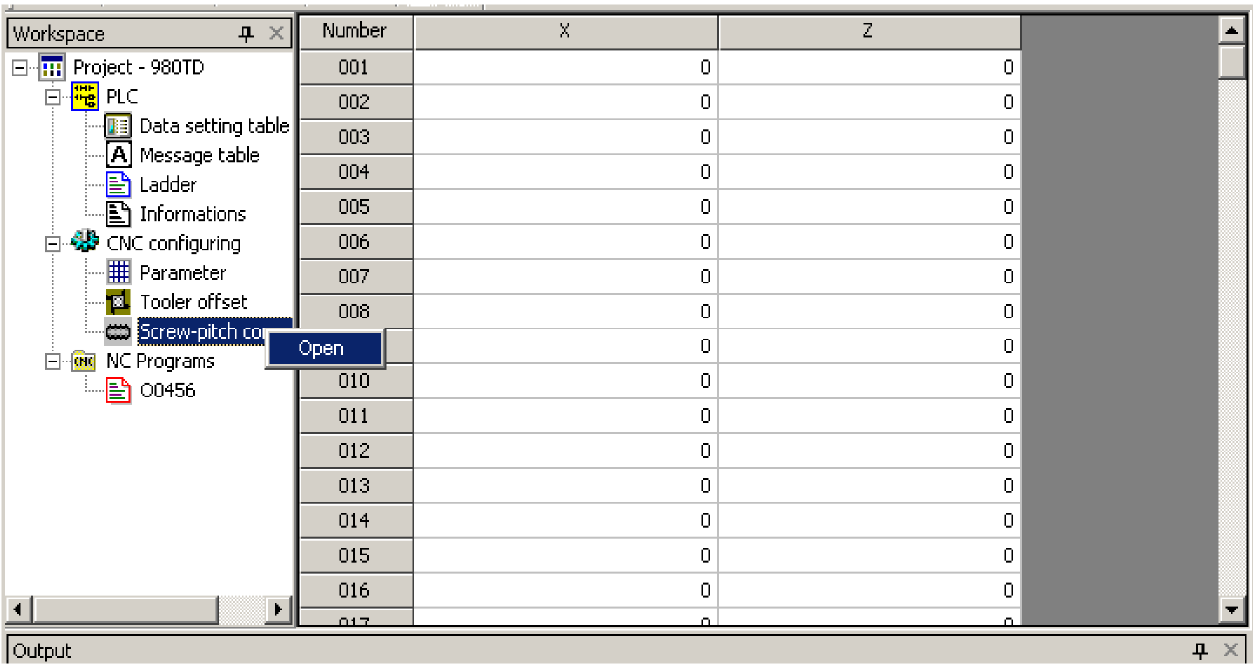


Fig. 4-15

4.9 PART PROGRAM

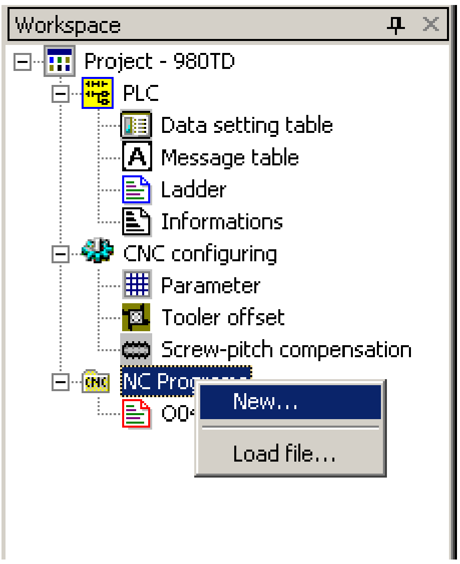


Fig. 4-16

As Fig. 4-16, right click [CNC Program] to pop-up：[New …] and [Load file]. New file is to create one CNC part program to project and Load file is to load a existed one on disk to project. Their methods are as follows: 1

4.9.1 Create a New NC Program

Pop-up a dialog box as Fig.4-17 after executing [New…] in Fig. 4-16, prompting to input program name(note: O is at program name's home and 1-4 digits are followed).

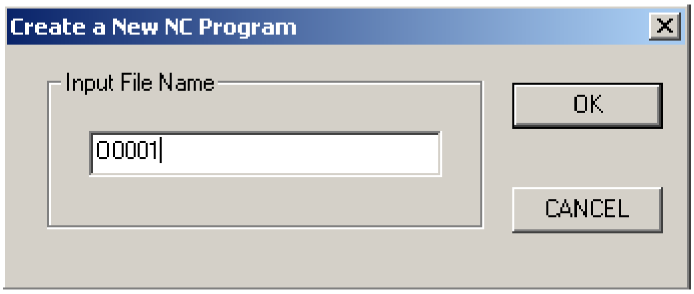


Fig. 4-17

After confirming CNC part program in the workspace, the created CNC part program is displayed and is opened to edit programs. Save them after editing the programs.

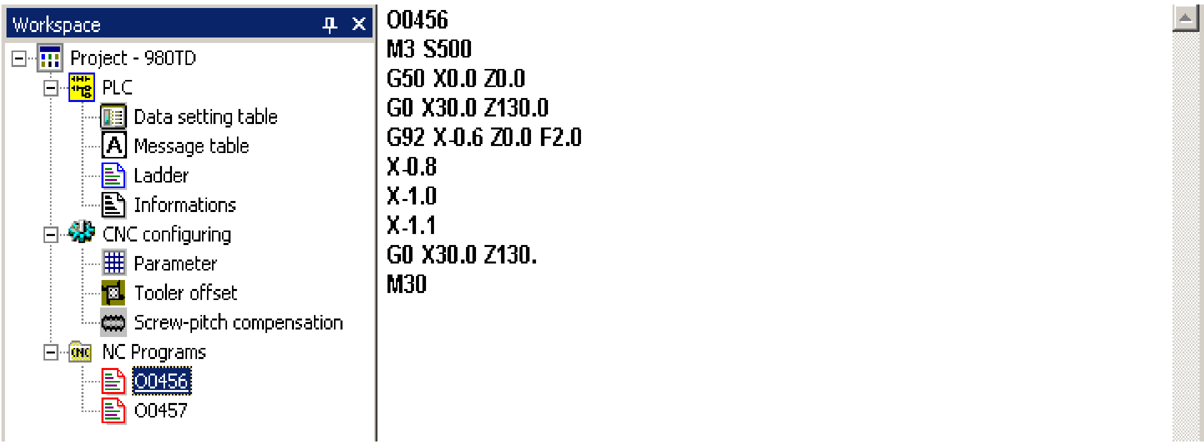


Fig. 4-18

4.9.2 Load to CNC Part Program

Pop-up a dialog box as Fig.4-19 after executing [Load File] in Fig. 4-16, and select the stored CNC part program on the disk..

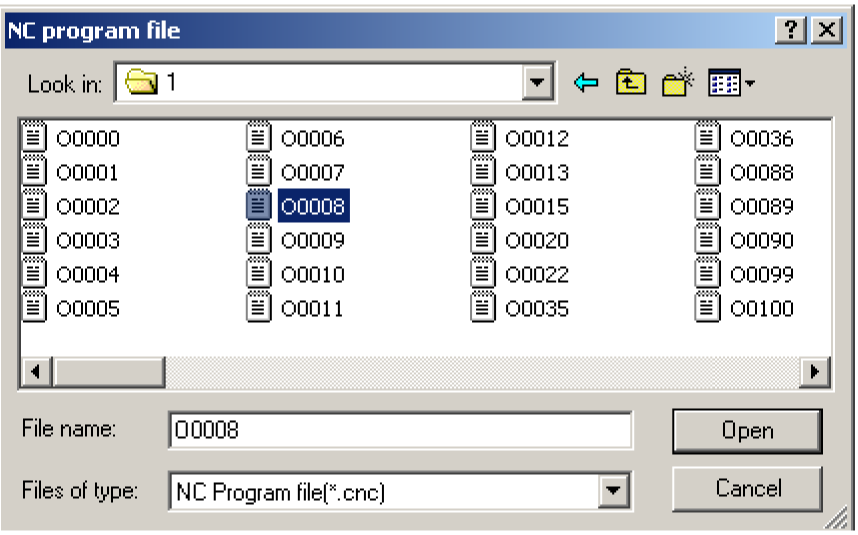


Fig. 4-19

Click [Open] to pop-up a dialog box as Fig.4-20:

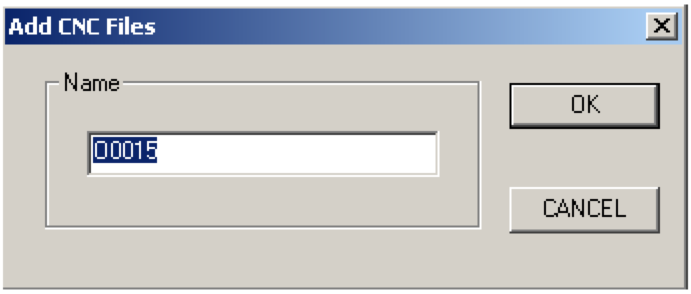


Fig. 4-20

After confirming the file name, load it to the current project. The system prompts to set a new program name again if there is the same one.

4.9.3 NC Part Program management

Manage part programs after operating the above methods, including modification, deletion. Open the corresponding program after double clicking program file menu as Fig. 4-20.

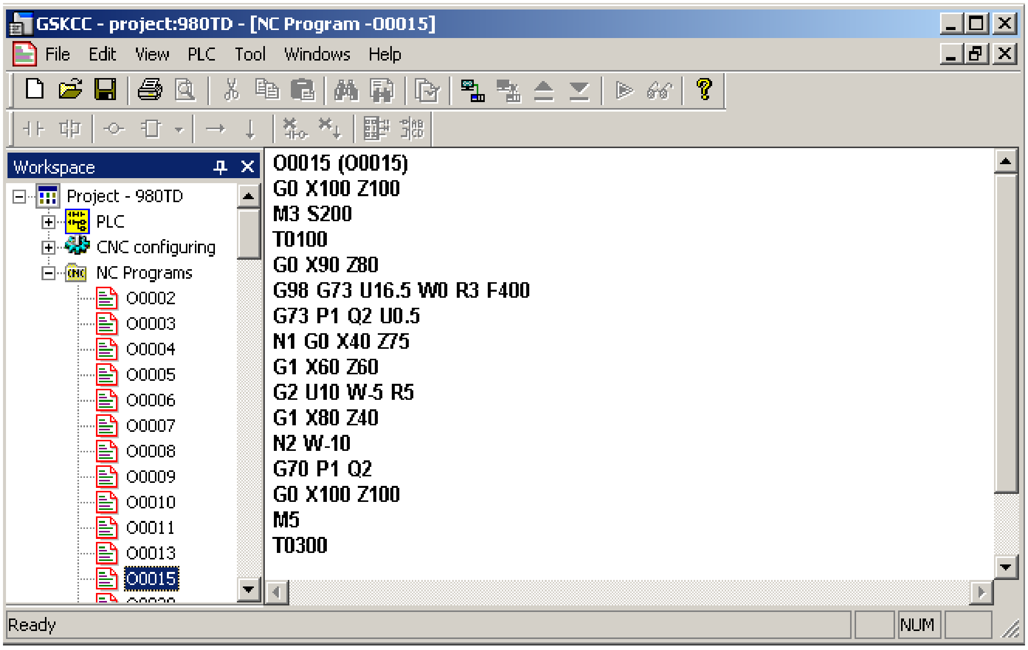


Fig. 4-21

Open programs or delete the unnecessary programs by right clicking menu as Fig. 4-22.

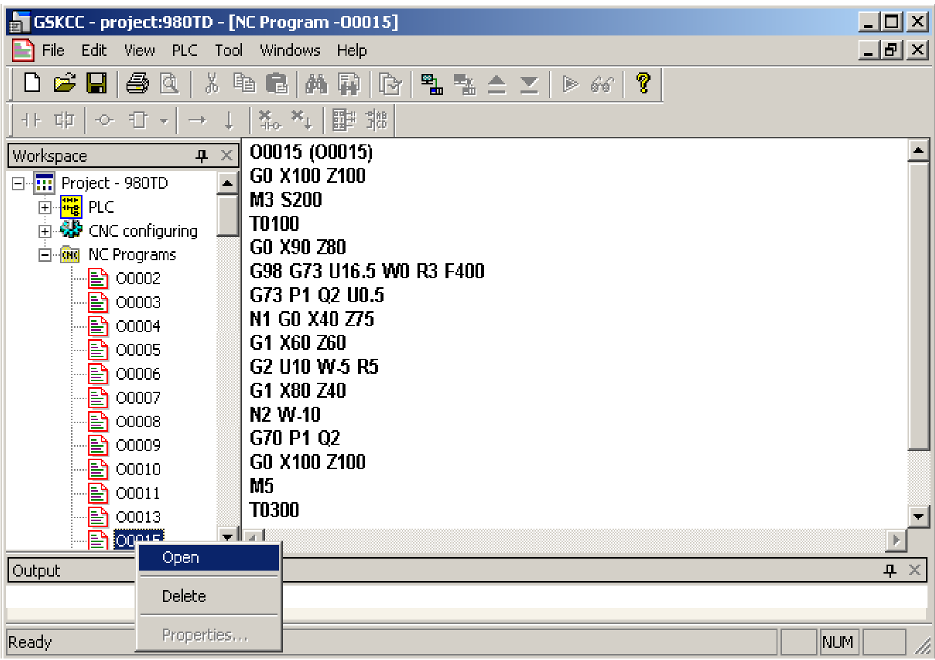


Fig. 4-22

Chapter 5 COMMUNICATION

GSKCC communicates with CNC by RS232,during which ladder of current project and parameters can be transmitted to CNC or ladder and parameters can be read from CNC

Rightly connect COMM port of PC with CNC communication interface, and set the corresponding parameters of GSKCC. Baud rate should be set same as that of CNC as follows:

Select [Tool]—[Communication Settings] to pop-up a dialog box (Fig. 5-2), and set the port and baud rate. Start the next operation after confirming.

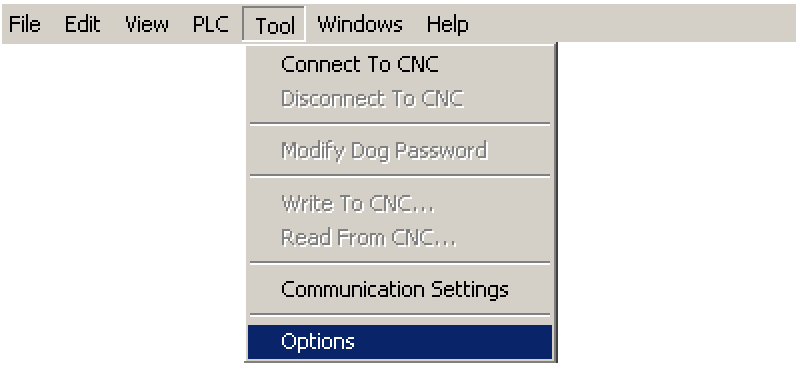


Fig. 5-1

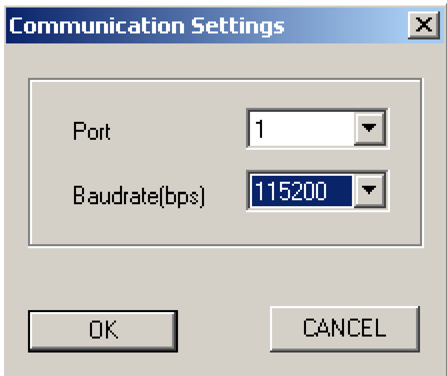
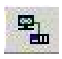


Fig. 5-2

5.1 TRANSMIT FILES to CNC

1. Connect with CNC

Run GSKCC and open the required project. Click  or [Tool]—[Connect to CNC] in Fig.5-1, and GSKCC prompts a dialog box [Connect To CNC successfully!]

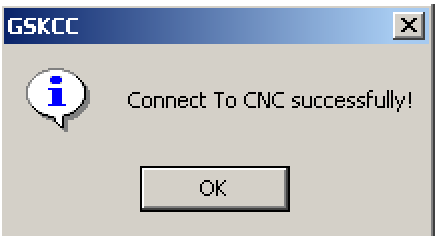
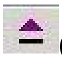


Fig. 5-3

After GSKCC is connected to CNC successfully, the toolbar is as follows: 连接成功后的工具条如下：



Fig. 5-4

(Disconnect with CNC), (Transfer current project to CNC), (Read Files from CNC) are in high light to execute the next operations.

2. File transmitting

Start to transmit files when CNC is in **Edit** mode,.

When the system transmits the screw-pitch compensation or PLC program, the user level of CNC is not less than 2, and the parameter switch is ON for transmitting;

When the system transmits the parameter, the user level of CNC is not less than 3 and the parameters switch is ON for transmitting;

When the system transmits tool offset, the user level of CNC is not less than 4 and the parameters switch is ON for transmitting;

When the system transmits part program, the user level of CNC is not less than 4 and the program switch is ON for transmitting. Press  to transmit files to pop-up a dialog box as follows:

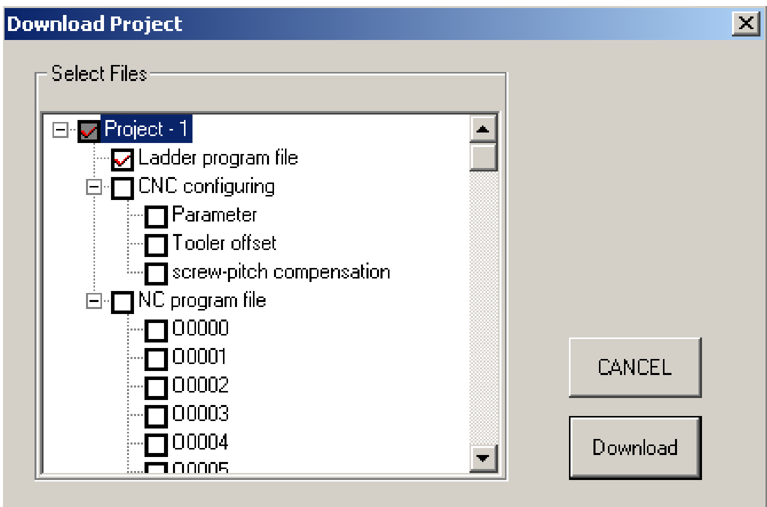


Fig. 5-5

Click the left box in [Download Project] to select the required files. The system selects files when there is a red √, otherwise, the system does not. Click [Download] to transmit files. There is a prompting box in transferring to display the current communication state, rate and so on.

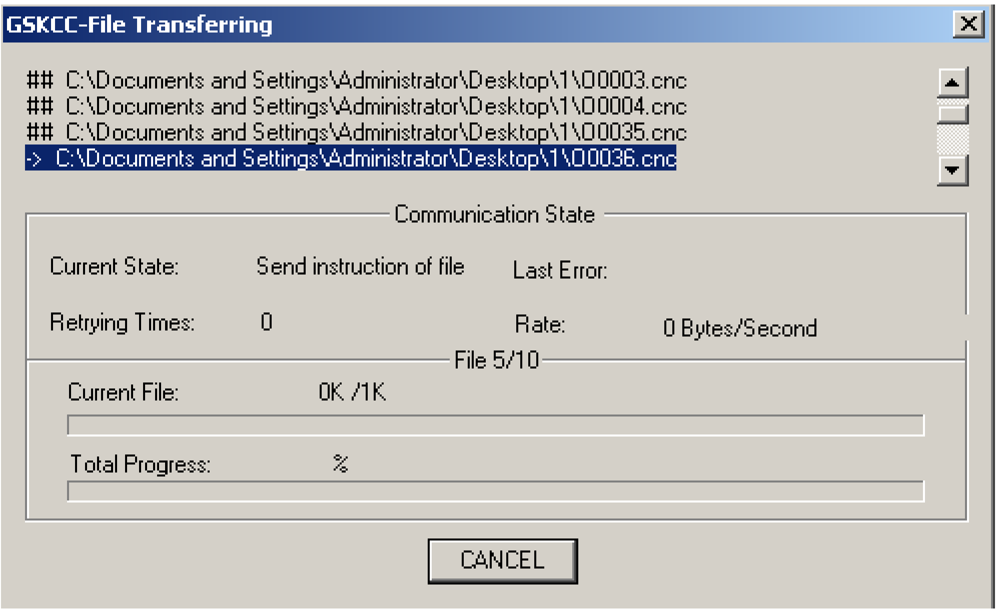


Fig. 5-6

5.2 UPLOAD FILES from CNC

Run GSKCC, and pop-up a dialog box after connecting to CNC and clicking .

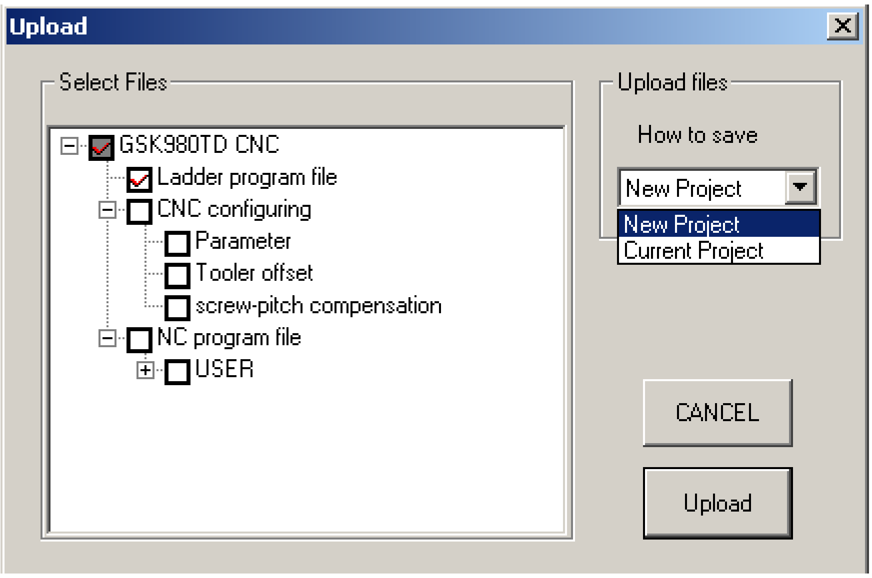


Fig. 5-7

For options pointed by cursor,[How to save] of [Upload files] can be selected:

a. **[New Project]:** If a project is opened currently, the system will close the current project and prompts the user to create a project to save the uploaded files when selecting [Upload files]; if a project is not opened, the system directly prompts to create a project to save the uploaded files.

b. **[Current Project]:** If a project is opened currently, one option is that ***<u>the current opened project is replaced by the upload file data</u>***, namely, the data of current project is replaced by uploading PLC programs, parameters from CNC.

After a file is selected in [Select Files], click [Upload] to upload the selected file to the project, and the corresponding setting view is automatically updated by the uploaded data.

5.3 DISCONNECT with CNC

After the communication is executed,  is clicked to disconnect with CNC in order that the serial is released or it is used by other application programs when GSKCC is connected to CNC.

BOOK APPENDIX

APPENDIX

APPENDIX 1: INPUT SINGAL（X）

X0								
	Input	Input	ESP	Input	XDEC	Input	Input	Input
Pin	XS39.12	XS39.11	XS40.10	XS40.22	XS40.1	XS40.2	XS40.3	XS40.4

X1								
	Input	Input	Input	Input	ZDEC	Input	Input	Input
Pin	XS40.5	XS40.6	XS40.19	XS40.7	XS40.9	XS40.21	XS40.8	XS40.20

X2								
	Input	Input	Input	Input	Input	Input	Input	Input
Pin	XS41.2	XS41.21	XS41.9	XS41.22	XS41.10	XS41.12	XS41.13	XS41.1

X3								
	Input	Input	Input	Input	Input	Input	Input	Input
Pin	XS41.4	XS41.6	XS41.5	XS41.19	XS41.7	XS41.20	XS41.8	XS41.3

X8								
				ESP				
Pin				XS40.10				

X20	7	6	5	4	3	2	1	0
Key	SKIP	SINGLE	JOG	MPG	MACHINE ZERO	MDI	AUTO	EDIT

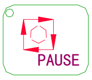
X21	7	6	5	4	3	2	1	0
Key	X	0.1	0.01	0.001	PROGRAM ZERO	DRY	MST M. S. T. LOCK	MST M. S. T. LOCK

X22	7	6	5	4	3	2	1	0
Key	Z		Z		X		Z	Y

X23	7	6	5	4	3	2	1	0
Key	CCW	JOG LUR.	STOP	COOLANT	CW		X	

X24	7	6	5	4	3	2	1	0
Key	PAUSE	%	%	%	%	%	%	TOOL

X25	7	6	5	4	3	2	1	0

Key								

X26	**7**	**6**	**5**	**4**	**3**	**2**	**1**	**0**
Key								

APEENDIX 2: OUTPUT SIGNAL (Y)

Address	Signal	Address	Signal
Y00.0	Universal output	Y01.0	Universal output
Y00.1	Universal output	Y01.1	Universal output
Y00.2	Universal output	Y01.2	Universal output
Y00.3	Universal output	Y01.3	Universal output
Y00.4	Universal output	Y01.4	Universal output
Y00.5	Universal output	Y01.5	Universal output
Y00.6	Universal output	Y01.6	Universal output
Y00.7	Universal output	Y01.7	Universal output
Y02.0	Universal output	Y03.0	Universal output
Y02.1	Universal output	Y03.1	Universal output
Y02.2	Universal output	Y03.2	Universal output
Y02.3	Universal output	Y03.3	Universal output
Y02.4	Universal output	Y03.4	Universal output
Y02.5	Universal output	Y03.5	Universal output
Y02.6	Universal output	Y03.6	Universal output
Y02.7	Universal output	Y03.7	Universal output
Y04.0	Auxiliary function locked indicator	Y05.0	Coolant indicator on machine control panel
Y04.1	Machine locked indicator	Y05.1	Spindle clockwise rotatition indicator on machine control panel
Y04.2	Optional block jump indicator	Y05.2	JOG mode indicator on machine control panel
Y04.3	Single block indicator	Y05.3	Handwheel/increment mode indicator on machine control panel
Y04.4	Rapid run indicator	Y05.4	Machine reference poin return mode indicator on machine control panel
Y04.5	End indicator for program reference point return/machine reference point return in Z direction	Y05.5	MDI mode indicator on machine control panel
Y04.6	End indicator for program reference point return/machine reference point return in Y direction	Y05.6	Auto mode indicator on machine control panel
Y04.7	End indicator for program reference point return/machine reference point return in X direction	Y05.7	Edit mode indicator on machine control panel

Y06.0	Spindle stopped indicator on machine control panel	Y07.0	Tool change indicator on machine control panel
Y06.1	Rapid feed indicator on machine control panel	Y07.1	Spindle counterclockwise rotation indicator on machine control panel
Y06.2	Program reference point return mode indicator on machine control panel	Y07.2	Handwheel indicator in Z direction on machine control panel
Y06.3	Dry run indicator on machine control	Y07.3	Handwheel indicator in Y direction on machine control panel
Y06.4	Auxiliary function locked indicator on mchine control panel	Y07.4	Handwheel indicator in X direction on machine control panel
Y06.5	Machine locked indicator on machine control panel	Y07.5	0.1 feedrate indicator on machine control panel
Y06.6	Optional block jump indicator on machine control panel	Y07.6	0.01 feedrate indicator on machine control panel
Y06.7	Single block indicator on machine control panel	Y07.7	0.001 feedrate indicator on machine control panel
Y08.0	Cycle start indicator on machine control panel	Y08.5	Min. rapid override indicator on machine control panel
Y08.1	Min.feedrate override indicator on machine control panel	Y08.6	Max. spindle override indicator on machine control light
Y08.2	Max. rapid override indicator on machine control panel	Y08.7	Spindle jog indicator on machine control panel
Y08.3	Min. spindle override indicator on machine control panel	Y09.0	Feed dwell indicator on machine control panel
Y08.4	Max.feedrate override indicator on machine control panel	Y09.1	Dry running indicator

APPENDIX 3: G, F SIGNAL

（Function sequence）

Function	Address	Signal name	Code	Chapter
Alarm signal	F1.0	Alarm signal	AL	2.3
Cancel override	G6.4	Override canceling signal	OVC	6.4
Feedrate override	G12	Feedrate override signal	FV0～FV7	6.3.1
	F210.0	External feedrate override switch input *0V1～*0V8 is valid	F_SOVI	12.7.1
Optional block jump signal	G44.0	Optional block jump signal	BDT	5.6
	F4.0	Detection signal for optional block jump	MBDT	
User macro program signal	G54,G55	User macro program input signal	UI0～UI15	10.1
	F54,F55	User macro program output signal	UO0～UO15	10.2
	F56～F59		U0100 ～ UO131	
	F210.6	Shield external cycle start signal (ST)	MST	12.7
	F210.5	Shield external pause signal(SP)	MSP	
	G7.2	Cycle start signal	ST	5.1
	G8.5	Feed pause signal	SP	
	F0.5	Cycle start alarm signal	STL	

	F0.4	Feed dwell alarm signal	SPL	
	F0.7	Automatic run signal	OP	
Function	**Address**	**Signal name**	**Code**	**Chapter**
Axis moving state output	F102.0～ F102.3	Axis moving signal	MV1～MV4	1.1
	F106.0～ F106.3	Axis motion direction	MVD1 ～ MVD4	
Spindle speed control	G29.4	Spindle speed in-position signal	SAR	8.4
	G30	Spindle override signal	SOV0 ～ SOV7	8.2.1
	G29.6	Spindle stop signal	*SSTP	
	G200.0	Spindle jog signal	SPHD	8.3
	F199.0	Spindle jog test signal	MSPHD	
	F200.4	Spindle mode signal	SIMSPL	8
	F208.3	Spindle enables the input singal is valid	SPEN	12.1.2
Spindle automatic gear shifting	F208.7	Valid signal of automatic gear shifting	AGER	8.2.2
	F208.6	Detection signal of gear shifting in-position when automatic gear shifting to 1, 2	AGIN	
Handwheel feed	G18.0, G18.1	Handwheel feed axis selection signal	HS1A,HS1B	3.2.1
	G19.4, G19.5	Handwheel/Incremental override signal	MP1,MP2	3.2.2
	F211.1	0.1mm increment selection signal on Handwheel/Incremental mode	SINC	12.8.6
Machine reference point	F4.5	Machine reference point return detection signal	MREF	2.5.2
	F201.5	Level selection for deceleration signal when executing machine reference point return	DECI	12.2
	F205.2	Self-hold of axis motion is valid in reference point return	ZNIK	
	F94.0～ F94.3	End signal of machine reference point return	ZP1～ZP4	4.1
	F120.0～ F120.3	Machine reference point creation signal	ZRF1 ～ ZRF4	
JOG feed/ incremental feed	G100	Selection signal of feed axis and direction	+J1～+J4	3.1.1
	G102		-J1～-J4	
	F214.0	X selection	XVAL	12.8
	F214.1	Z selection	ZVAL	
JOG feed/ incremental feed	G10,G11	Manual traverse override signal	JV0～JV15	3.1.2
	G19.7	Manual rapid feed selection signal	RT	3.1.3
CNC ready signal	F1.7	CNC ready signal	MA	2.2
	F0.6	Servo ready signal	SA	1.2
State output signal	F2.1	Rapid feed signal	RPDO	2.6
	F2.6	Cutting feed signal	CUT	
	F1.5	Tapping signal	TAP	
	F2.3	Thread cutting	THRD	
Single block	G46.1	Single block signal	SBK	5.5
	F4.3	Single block test signal	MSBK	
Start locked	G7.1	Start locked signal	STLK	2.4

Dry run	G46.7	Dry run signal	DRN	5.4
	F2.7	Dry run test signal	MDRN	
Function	**Address**	**Signal name**	**Code**	**Chapter**
Rapid feedrate override	G14.0, G14.1	Rapid feedrate override signal	ROV1,ROV2	6.2.2
Emergent stop	X0.5	Emergent stop signal	ESP	2.1
	G8.4			
Auxiliary function	F10～F13	Auxiliary function code signal	M00～M31	7.1.1
	F7.0	Auxiliary function strobe signal	MF	
	F9.4	M decode signal	DM30	7.1.2
	F9.5		DM02	
	F9.6		DM01	
	F9.7		DM00	
	F22～F25	Spindle speed code signal	S00～S31	7.2
	F7.2	Spindle speed strobe signal	SF	
	F26～F29	Tool function code signal	T00～T31	7.3
	F7.3	Tool function strobe signal	TF	
	G4.3	End signal of auxiliary function	FIN	7.4.1
	F1.3	End signal of distribution	DEN	7.4.2
Tool change function	G44.7	Manual sequential tool change signal	HDT	9.2
	F205.0	Level selection for toolpost locked signal	TCPS	12.3
	F205.1	Level selection for toolpost in-position signal	TSGN	
	F207	Total tool selection	TMAX	
	F223.0	Tool change mode selection	PB5	
	F223.2	If in-position of tool change end is tested	PB6	
Auxiliary function locked	G5.6	Auxiliary function locked signal	AFL	7.5
	F4.4	Auxiliary function locked test signal	MAFL	
Machine locked	G44.1	All axes on machine locked signal	MlK	5.3
	F4.1	All axes on machine locked test signal	MMLK	
Soft operator panel	F75.1	Auxiliary function locked	AFLO	11.1
	F75.2	Program jump	BDTO	
	F75.3	Single block	SBKO	
	F75.4	Machine locked	MLKO	
	F75.5	Dry run	DRNO	
	F197.1	OUT key	OUTO	
Memory defence	G46.3	Memory defence signal	KEY1	11.2
Mode selection	F3.0	Increment mode detection signal	MINC	2.5
	F3.1	Handwheel mode detection signal	MH	
	F3.2	JOG mode detection signal	MJ	
	F3.3	MDI mode detection signal	MMDI	
	F3.5	Auto mode detection signal	MMEM	
	F3.6	Edit mode detection signal	MEDT	
	F4.5	Machine reference point return detection mode signal	MREF	

	F4.6	Program reference point return detection mode signal	MPRO	
	G43.0～G43.2, G43.5, G43.7	Mode selection signa	MD1,MD2, MD4 DNC1,ZRN	
Rigid tapping	G74.4	Signal of spindle clockwise rotation in rigid tapping	SRVB	8.5
	G74.5	Signal of spindle counterclockwise rotation in rigid tapping	SFRB	
	F76.3	Signal in rigid tapping	RTAP	
Reset	G8.7	External reset signal	ERS	5.2
	F1.1	Reset signal	RST	
Selection signal with reset key for output interface	F204.3	Selection signal with reset key for output interface	RSJG	12.7
Tailstock control	F208.2	Tailstock control is invalid/valid	SLTW	12.8.3
Internal/external chuck function	F208.0	Chuck function is valid	SLSP	12.4
	F208.1	Do not check chuck signal	SLQP	
	F209.0	Internal chuck selection	PB1	
	F209.2	Chuck in-position signal test is valid	PB2	
Pressure low detection	F209.4	Pressure low detection is valid	PB3	12.5
	F209.5	Level selection for pressure signal	SPB3	
Safeguard detection	F209.6	Safeguard detection is valid	PB4	12.6
	F209.7	Level selection for safeguard input signal	SPB4	
S1-S4 output selection	F211.0	S1-S4 output selection	SUOS	12.1.6

（Address sequence）

Function	Address	Signal name	Code
G4.3	Signal for auxiliary function end	FIN	
G5.6	Signal for auxiliary function locked	AFL	
G6.4	Override canceling signal	OVC	
G7.2	Cycel start signal	ST	
G8.0	Interlock signal	IT	
G8.4	Emergent stop signal	ESP	
G8.5	Feed pause signal	SP	
G8.7	External reset signal	ERS	
G10,G11	Manual traverse override signal	JV0～JV15	
G12	Feedrate override signal	FV0～FV7	
G14.0, G14.1	Rapid feedrate override signal	ROV1, ROV2	
G18.0, G18.1	Handwheel feed axis selection signal	HS1A, HS1B	
G19.4, G19.5	Handwheel/Incremental override signal	MP1, MP2	
G19.7	Manual rapid feed selection signal	RT	
G29.4	Spindle speed in-position signal	SAR	
G29.6	Spindle stop signal	*SSTP	

Function	Address	Signal name	Code
G30	Spindle override signal	SOV0～SOV7	
G43.0～G43.2, G43.5, G43.7	Mode selection signal	MD1, MD2, MD4, DNC1, ZRN	
G44.0	Optional block jump signal	BDT1	
G44.1	All axes on machine locked signal	MlK	
G44.7	Manually sequential tool change signal	HDT	
G46.1	Block signal	SBK	
G46.3	Memory defence signal	KEY1	
G46.7	Dry run signal	DRN	
G54, G55	User maro program input signal	UI0～UI15	
G74.4	Spindle clockwise rotation in rigid tapping	SRVB	
G74.5	Spindle counterclockwise rotation in rigid tapping	SFRB	
G100	Feed axis and direction selection signal	+J1～+J4	
G102		-J1～-J4	
G114	Overtravel signal	+L1～+L4	
G116		-L1～-L4	
G200.0	Spindle jog function signal	SPHD	
G201～G204	Current tool selection signal		
F0.4	Feed dwell lamp signal	SPL	
F0.5	Cycle start lamp signal	STL	
F0.6	Servo ready signal	SA	
F0.7	Automatic run signal	OP	
F1.0	Alarm signal	AL	
F1.1	Reset signal	RST	
F1.3	Distribution end signal	DEN	
F1.5	Tapping signal	TAP	
F1.7	CNC ready signal CNC	MA	
F2.1	Rapid feed signal	RPDO	
F2.3	Thread cutting	THRD	
F2.6	Cutting feed signal	CUT	
F2.7	Dry run test signal	MDRN	
F3.0	Increment mode detection signal	MINC	
F3.1	Handwheel mode detection signal	MH	
F3.2	JOG mode detection signal	MJ	
F3.3	MDI mode detection signal	MMDI	
F3.5	Auto mode detection signal	MMEM	
F3.6	Edit mode detection signal	MEDT	
F4.0	Optional block jump detection signal	MBDT1	
F4.1	All axes on machine locked detection signal	MMLK	
F4.3	Single block detection signal	MSBK	
F4.4	Detection signal for auxiliary function locked	MAFL	
F4.5	Detection signal for machine reference point return mode	MREF	
F4.6	Detection signal for program reference point return mode	MPST	
F7.0	Auxiliary function strobe signal	MF	

Function	Address	Signal name	Code
F7.2	Spindle speed function strobe siganl	SF	
F7.3	Tool function strobe signal	TF	
F9.4	M encode signal	DM30	
F9.5		DM02	
F9.6		DM01	
F9.7		DM00	
F10～F13	Auxiliary function code signal	M00～M31	
F22～F25	Spindle speed code signal	S00～S31	
F26～F29	Tool function code signal	T00～T31	
F53.4	Background busy signal	BGEACT	
F54、F55	User macro program output signal	UO0～UO15	
F56～F59		U0100～UO131	
F75.1	Auxiliary function locked soft key	AFLO	
F75.2	Program optional jump soft key	BDTO	
F75.3	Block soft key	SBKO	
F75.4	Machine locked soft key	MLKO	
F75.5	Dry run soft key	DRNO	
F76.3	Rigid tapping process signal	RTAP	
F94.0～ F94.3	End signal of machine reference point return	ZP1～ZP4	
F102.0～ F102.3	Axis traversing signal	MV1～MV4	
F106.0～ F106.3	Axis motion direction signal	MVD1～MVD4	
F120.0～ F120.3	Reference point creation signal	ZRF1～ZRF4	
F197.1	OUT key in MDI mode MDI		
F198.0～ F198.3	End signal of program reference point return	PRO1～PRO4	
F199.0	Spindle jog detection signal	MSPHD	
F200.4	Analog spindle is valid	SIMSPL	NO.001.4
F201.2	If OUT key starts programs (only in MDI mode)	DCS	NO.004.2
F201.5	Level selection for deceleration signal when executing reference point return	DECI	NO.004.5
F204.3	Selection Signal with Reset key for output interface	RSJG	NO.009.3
F205.0	Level selection for toolpost locked signal	TCPS	NO.011.0
F205.1	Level selection for toolpost in-position signal	TSGN	NO.011.1
F205.2	Self-hold of axis motion is valid in reference point return	ZNIK	NO.011.2
F207	Total tool selection	TMAX	NO.084
F208.0	Chuck function is valid	SLQP	NO.164.0
F208.1	Do not check chuck clamping signal	SLSP	NO.164.1
F208.2	Tailstock control function is invalid/valid	SLTW	NO.164.2
F208.3	Spindle rotating enables the input signal is valid	SPEN	NO.164.3
F208.5	Level selection for gear shifting in-positon signal	AGIM	NO.164.5
F208.6	Check gear shifting in-position signal when gear shifting to 1, 2	AGIN	NO.164.6
F208.7	Automatic gear shifting is valid	AGER	NO.164.7
F209.0	Chuck mode selection	PB1	NO.168.0
F209.2	Detection function for chuck in-position signal is valid	PB2	NO.168.2
F209.4	Pressure low detection is valid	PB3	NO.168.4

Function	Address	Signal name	Code
F209.5	Level selection for pressure signal	SPB3	NO.168.5
F209.6	Safteguard detection function is valid	PB4	NO.168.6
F209.7	Level selection for safeguard input signal	SPB4	NO.168.7
F210.0	Input *0V1～*0V8 for external feedrate override switch is valid	SOVI	NO.172.0
F210.3	Shield the external emergent stop signal(*ESP)	ESP	NO.172.3
F210.5	Shield the external pause signal(*SP)	MSP	NO.172.5
F210.6	Shield the external cycle start signal(ST)	MST	NO.172.6
F211.0	S1-S4 output selection	SUOS	NO.173.0
F211.1	0.1mm feedrate selection signal on Handwheel/Incremental mode	SINC	NO.173.1
F212.2	The system is executing G04(dwell)	CDWL	
F212.3	The system is executing check	CINP	
F212.4	Override of 0%	COVL	
F213.0	Outputtingm message by RS232 interface	CTPU	
F213.1	Inputting message by RS232 interface	CTRD	
F214.0	Select X direction	XVAL	NO.175.0
F214.1	Select Z direction	ZVAL	NO.175.1
F214.7	Spindle jog/lubrication		NO.175.7
F223.0	Select a tool change mode	PB5	NO.182.0
F223.2	Whether the in-position signal is detected at the end of tool changing	PB6	NO.182.2

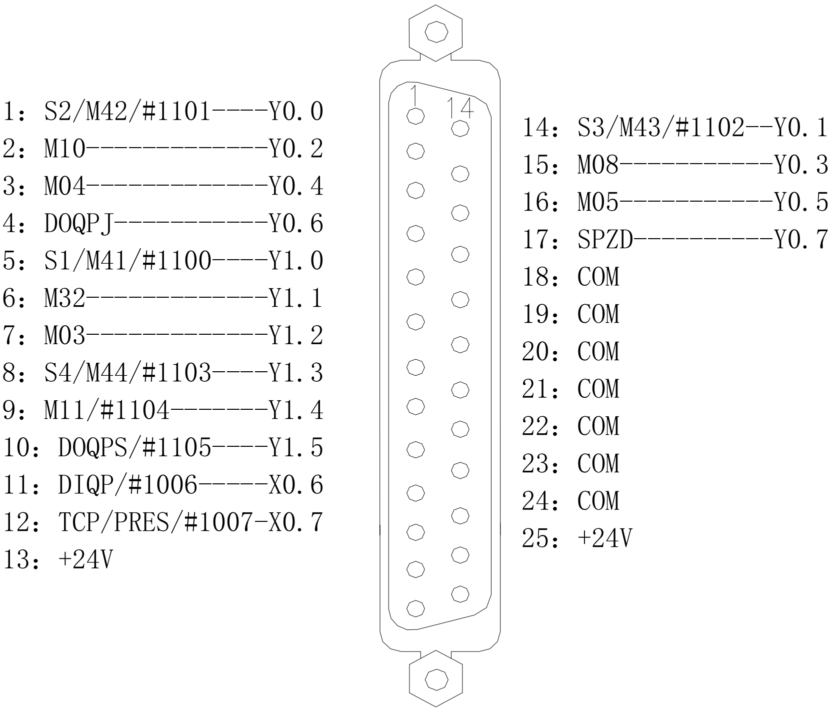
APPENDIX 4: GSK980TD STANDARD FUNCTION CONFIGURATION

A.DT address definition

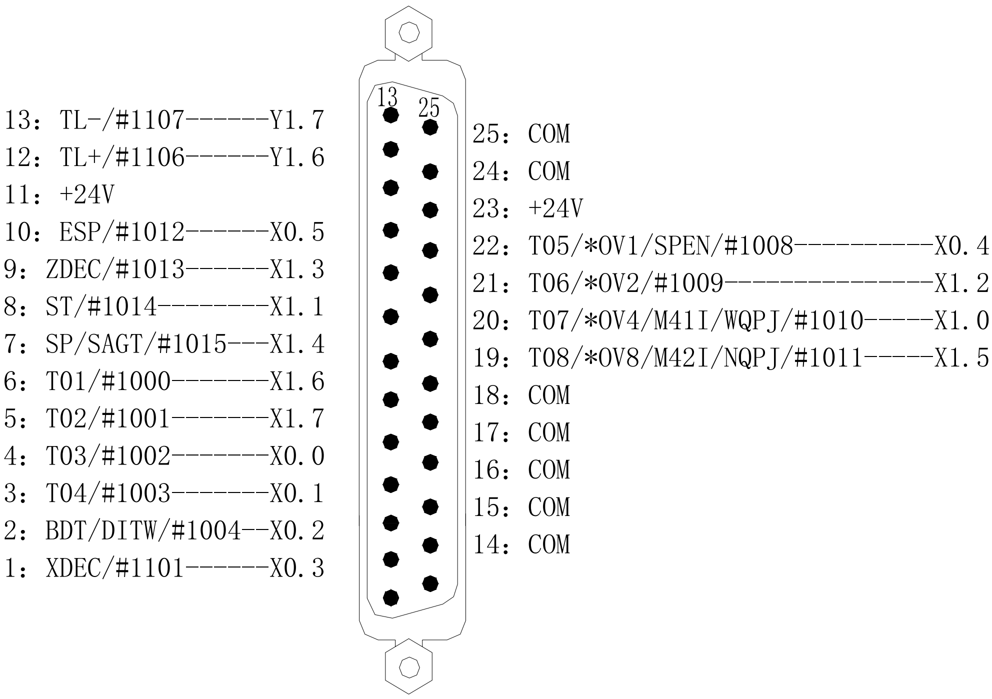
Address	Definition	Parameter
DT0000	Gear shifting time 1	NO.065
DT0001	Gear shifting time 2	NO.066
DT0002	Alarm time width for low pressure	NO.069
DT0003	Time upper limit for moving one tool when executing tool change	NO.076
DT0004	Time upper limit for tool change from the first tool to the last one	NO.078
DT0005	Executive duration for M code	NO.080
DT0006	Executive duration for S code	NO.081
DT0007	Delay from stop of toolpost clockwise rotation to start of counterclockwise locked	NO.082
DT0008	Alarm time without receiving*TCP	NO.083
DT0009	Locked time for toolpost counterclockwise rotation	NO.085
DT0010	Output time from spindle stop to braking	NO.087
DT0011	Spindle braking output time	NO.089
DT0012	Spindle rotational time when spindle jogging	NO.108
DT0013	Lubrication output pulse time	NO.112
DT0016	Automatic lubrication meantime	
DT0017	Automatic lubrication output time	
DT0018	Chuck pulse output width	

DT0021	Spindle stop, chuck operation enabling delay	

B. X, Y address definition



XS39 output



XS40 input

C. GSK980TD standard function ladder

广州数控设备有限公司
GSK CNC EQUIPMENT CO., LTD.

Add: No.52, 1st . Street, Luochong North Road, Luochongwei, Guangzhou, 510165, China

Website: http://www.gsk.com.cn E-mail: sale1@gsk.com.cn

Tel: 86-20-81796410/81797922 Fax: 86-20-81993683

All specification and designs are subject to change without notice July. 2006/Edition 2

Suit for software version:V06.03.23 July. 2006/Printing 2